INTERGRAPH®
Smart 3D

Hangers and Supports
User's Guide
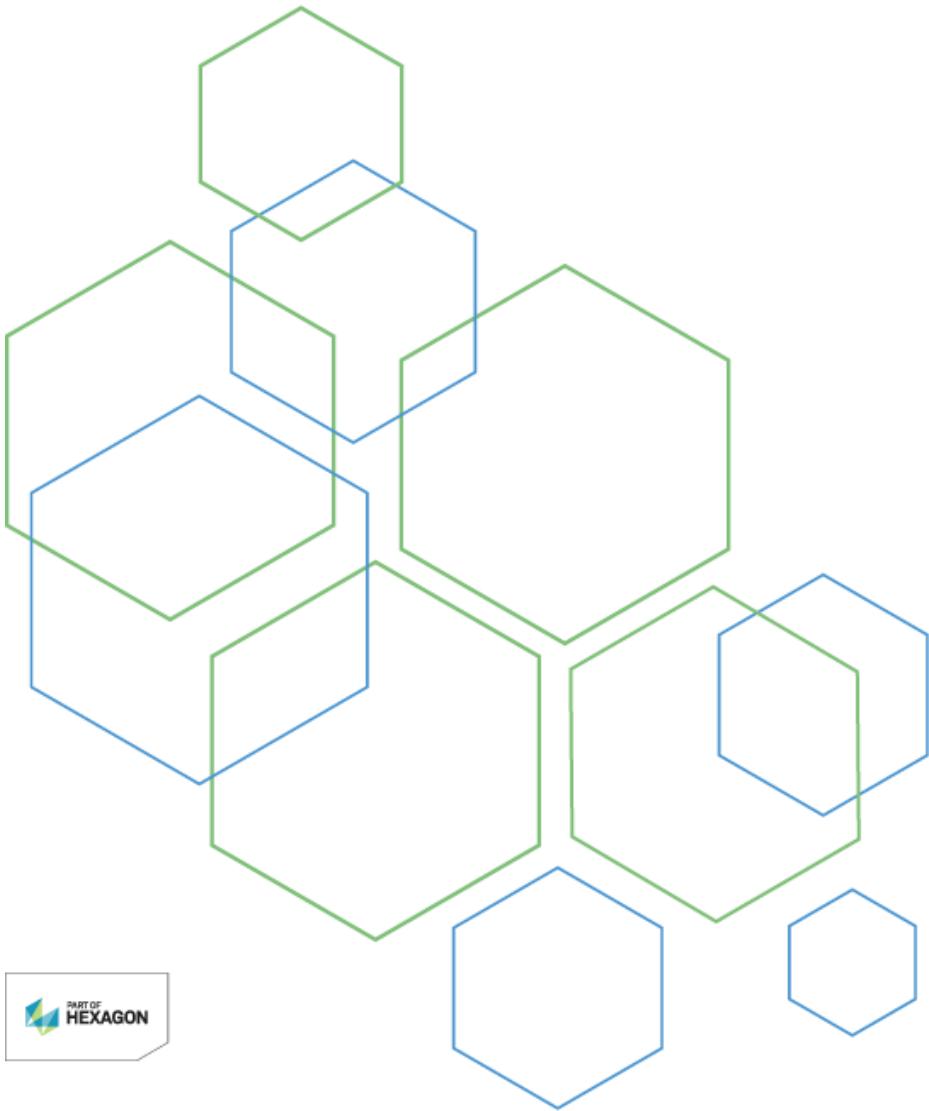
PART OF
HEXAGON

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Contents

Preface

This document is a user's guide for the Hangers and Supports functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Hangers and Supports

The following changes have been made to the Hangers and Supports task.

Version 2016 (11.0)

- Added a new locate filter, **Construction Graphics**. For more information, see *Edit Supports* (on page 12). (P2 CP:271166)

- Added a new custom command, **Placing Supports from XLS**. For more information, see *Placing Supports from XLS Command* (on page 175). (P2 CP:284643)

- You can now place hangers by selecting civil trenches as supporting objects. For more information, see *Hangers and Supports Workflow (on page 11).* (P3 CP:266512)

Hangers and Supports

The primary purpose of hangers and supports is to support various types of distributive systems such as pipes, HVAC ducts, and cable trays. Usually, supports are connected to the supported object, such as a pipe, and to a supporting object, such as a beam. However, you can also place supports between two or more supported objects such as placing a support between two pipes.

NOTES

- Throughout this document, the word "support" is used as a generic way to refer to both supports and hangers.

- Throughout this document, the word "feature" is used as a generic way to refer to pipes, HVAC ducts, cable trays, cable ways, and conduit.

The Hangers and Supports task treats supports as first class modeling components. The supports are actually connected to piping and structural systems instead of simple graphical references. Connections enable the supports to automatically react to design changes. For example, if the size of a pipe changes, the pipe supports automatically react. Additionally, popular support catalogs are included with the software. You can also model designed supports.

The Hangers and Supports task has these commands:

Select - Selects objects in the model. For more information, see *Edit Supports* (on page 12).

Place Support by Structure - Places a support at the intersection of a supporting object and the supported object. For more information, see *Place Support by Structure* (on page 15).

Place Support by Point - Places a support at a location that you specify. For more information, see *Place Support by Point* (on page 22).

Place Support by Reference - Places a support at a location that you specify with a reference point as structure reference, see *Place Support by Reference* (on page 41).

Place Part - Adds or modifies parts in a designed support. For more information, see *Place Part* (on page 46).

Drop Standard - Converts a standard support to a designed support. For more information, see *Drop Standard* (on page 68).

Start 3rd Party App - Starts a third-party application to add parts to a designed support that you have already placed in the model. For more information, see *Start 3rd Party App* (on page 74).

Place Linear Member - Places columns, beams, braces, and other linear members in the model. For more information, see *Place Linear Member Systems* (on page 77).

Place Member Assembly - Places assembly connections between linear member

systems. For more information, see *Place Assembly Connection* (on page 152).

Trim Member - Manually copes and snips member parts based on objects and planes that you specify. For more information, see *Trim Members* (on page 169).

S ECTION 2

Hangers and Supports Workflow

Hangers and supports are placed in the model using information defined in the hangers and supports reference data. Using the reference data workbook, you can create parts, define assemblies, and define placement rules. Your first step is to review, edit, and otherwise customize the delivered hangers and supports reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the Smart 3D software.

After the reference data is customized to suit your needs, go to the Systems and Specifications task and define the systems that you want in your model. While you are not required to create your systems first, doing so keeps you from having to edit your hangers and supports after placement to assign them to the correct system.

Because hangers and supports are dependent on the optional supporting object and the supported object, you need to have those objects placed in the model before placing any hangers or supports. Examples of supporting objects include beams, plates, slabs, equipment, walls, and trenches. Examples of supported objects include piping, HVAC, conduit, and cable trays. These objects are placed using the Piping, HVAC, and Electrical tasks.

See Also

Place multiple supports along a feature (on page 29)
Place a support at a specific location (on page 27)
Place a support by structure (on page 19)
Design a support by point (on page 49)
Design a support by structure (on page 49)
Hangers and Supports (on page 9)

Hangers and Supports Common Tasks

The following tasks are used frequently in the Hangers and Supports task.

Customize Reference Data

Create new hangers and supports by editing the applicable workbooks. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Placing Supports

Place a hanger from a specific structure. For more information, see *Place a Support by Structure* (on page 19).

Place a hanger or support at a specific location. For more information, see *Place a support at a specific location* (on page 27).

Place multiple hangers or supports along a run. For more information, see *Place multiple supports along a feature* (on page 29).

Edit Supports

All objects in the Hangers and Supports task have properties that you can edit. Using the **Select** command on the vertical toolbar, select the object that you want to edit.

An important part of the **Select** command is the **Locate Filter** list that appears on the ribbon. The **Locate Filter** list contains the available, pre-defined filters for the **Select** command. When you select a filter in the **Locate Filter** list, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Support**, you can select only supports in a graphic view or in the **Workspace Explorer**.

The Hangers and Supports task includes these filters:

Assembly Components

> Limits your selection in a graphic view or in the **Workspace Explorer** to assembly components.

Construction Graphics

> Limits the selection of items to construction graphics.

Control Points

> Limits your selection in a graphic view or in **Workspace Explorer** to control points.

Frame Connections

> Limits your selection in a graphic view or in the **Workspace Explorer** to frame connections.

Member Assembly Connections

> Limits your selection in a graphic view or in the **Workspace Explorer** to member assembly connections.

Member Parts

> Limits your selection in a graphic view or in the **Workspace Explorer** to member parts.

Member Systems

> Limits your selection in a graphic view or in the **Workspace Explorer** to member systems.

Support

> Allows you to select supports in a graphic view or in the **Workspace Explorer**.

Support Component

> Limits your selection in a graphic view or in the **Workspace Explorer** to the individual parts of a support system.

Support Connection

> Limits your selection in a graphic view or in the **Workspace Explorer** to a support connection.

All

Allows you to select any object, even objects created in another task.

 Use the **Inside** fence command to select all objects entirely inside the fence.

 Use the **Inside/Overlapping** fence command to select all objects entirely inside the fence and those objects outside but touching the fence at some point.

NOTE When you select multiple supports, the software highlights the first selected support in cyan. You can easily see which support is the origin of the spacing as you space multiple supports along a feature.

Naming Rules

The software provides several options for naming Hangers and Supports that you create. These naming rules are listed in the **GenericNameRules.xls** spreadsheet located in the appropriate install folder: ...\CatalogData\BulkLoad\DataFiles. For more information on creating naming rules, see the *Smart 3D Reference Data Guide* available using the **Help > Printable Guides** command in the software.

User Defined

Allows you to define a custom name for the support component. Type the name of the Support component in the **Name** box.

Tag Name Rule

For parent assemblies, the naming rule creates a name based on the part or the assembly being used, Global Workshare location ID, and an index number. The naming rule inserts a dash "-" between each section of the name. For example, a parent assembly placed on a pipeline is called Assy_FR_LS_CS_1-1-1.

For child parts of the assembly, the naming rule creates a name based on the part or assembly being used, Global Workshare location ID, and an index number preceded by a "C." The naming rule inserts a dash "-" between each section of the name. For example, a child part used by an assembly is called HgrAisc31_C3x6-1-C1. If more than one of a particular part is used in an assembly, the index number increases. For example, a second part is called HgrAisc31_C3x6-1-C2.

The following is an example of three different assemblies placed on the same pipeline.

```
2001-P
  Assy_FR_LS_CS_1-1-1
      HgrAisc31_C3x6-1-C1
      HgrAisc31_C3x6-1-C2
      Utility_FOUR_HOLE_PLATE_4-1-C111
  Assy_FR_LS_LS_3-1-1
      HgrAisc31_L3X3X1/4-1-C1
      HgrAisc31_L3X3X1/4-1-C2
      Utility_FOUR_HOLE_PLATE_4-1-C101
  Assy_RR_DR_WS_11-1-1
      Anvil_FIG140_4-1-C2
      Anvil_FIG140_4-1-C3
      Anvil_FIG290_4-1-C2
      Anvil_FIG290_4-1-C3
      Anvil_FIG299_4-1-C1
      Anvil_FIG299_4-1-C2
      Anvil_FIG55S_4-1-C2
      Anvil_FIG55S_4-1-C3
      Anvil_FIG66_4-1-C1
      Anvil_FIG66_4-1-C2
      Anvil_HEX_NUT_4-1-C3
      Anvil_HEX_NUT_4-1-C4
      HgrAisc31_W4X13-1-C1
```

S E C T I O N 3

Place Support by Structure

Places a support at the intersection of a selected feature and a selected structure. The software automatically determines the intersection point for you. Use this command when you want to support a feature that is not parallel to the supporting structure.
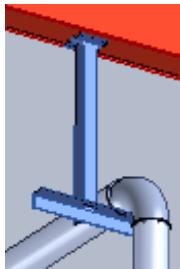
Support Placement by Structure Ribbon

Sets options for placing a support in the model.

Properties

Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box* (on page 32).

Custom Form Definition

Opens the user defined form for an object that has a form defined. It is enabled only if a proper form definition is defined for that particular object. For the remaining objects, it is disabled. For more information, see User-defined Forms.

Feature

Select the features for which you want to place a support. You must select at least one feature. Click **Reject** ✕ to clear all the selected features and start selecting the features again. Click **Accept** ✓ when you are finished selecting the features.

If you are placing the support using **Place Support by Point** , the first feature that you select is the primary feature. The system uses the cross section from the primary feature to define the support plane. If you clear the primary feature, the second feature that you select becomes the new primary feature.

Structure

Select the structure to support the feature you selected. Click **Reject** to clear all selected structures and start selecting supporting structures. Click **Accept** when you are finished selecting supporting structures.

Finish

Places the support using the information that you provided.

✕ **Reject**

Clears all the selected objects and restarts the selection process.

✓ **Accept**

Indicates that the object selection is complete.

System

Select the parent system for the support that you are placing. You can create systems in the Systems and Specifications task.

Rule

Displays the support types based on the rules defined in the reference data. When you select this option, the software uses an Assembly Selection Rule to filter the available supports in the **Type** box based on parameters such as the pipe outside diameter, distance between the supported object and the supporting object, and whether the route is above or below the supporting object. You or your project administrator have control over how the rules are defined. See the *Hangers and Supports Reference Data Guide* for more information. Clear the **Rule** option to select support types that are available.

Type

Sets the kind of support that you want to place. If you select the **Rule** option, the software lists support types based on the rules defined in the reference data. If you clear the **Rule** option, the software lists the last ten support types that you placed. Select **More** to select a different support type.

TIP When you place a designed support, the **Type** box is not available.

⬚ **Change Support Configuration**

Cycles through the available attachment types based on the feature and the support type. This option is not available for types that do not support multiple attachment types.

⚍ **Toggle Connection to Structure**

Cycles through the available attachment types based on the supporting structure and the support type. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option is not available for types that do not support multiple attachment types.

⊤ **Toggle Face Position**

Cycles through the available face positions for the supporting structure. For example, you can use this option to move a U-bolt so that it does not interfere with the web of a beam. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option might not be available if there are no optional face positions.

Design

Places the support as a Design support. Select the **Design** check box, when starting the placement command, to place a Design support. Clear the **Design** check box before committing the selection of the feature, to place a Standard assembly. You can select or clear the **Design** check box any time when the Straight Feature SmartStep is active. You can even select or clear it when returning after committing the feature selection with the Green Check. If you select **Design**, the **Rule** check box is disabled.

NOTE The parts you can use to create a design support are available in the catalog. These

parts are intelligent and contain property information that enables you to generate reports for the entire support.

Select Support Dialog Box

Allows selection of the type of support to be placed. This dialog box appears automatically when you select the **More...** option in the **Type** box on the ribbon. By browsing through the part hierarchy, you can find any support in the Catalog database. After you select a support, the software returns you to the model, where you can finalize placement.

Back

Returns you to the previously selected support type or node. Use this command to navigate through the support hierarchy to the specific type you need.

Forward

Sends you to the last selected support type or node that you moved away from by using the **Back** button. Use this command to navigate through the support hierarchy to the specific type you need.

Up One Level

Brings up the next highest level of the support catalog hierarchy. Use this command to navigate through the support hierarchy to the specific type you need.

Copy

Copies the selected object. This command is available only in the Catalog task.

Paste

Pastes a copied object. This command is available only in the Catalog task.

Delete

Deletes the selected object. This command is available only in the Catalog task.

Undo

Reverses the most recent operation. This command is available only in the Catalog task.

New Object

Creates a new object. This command is available only in the Catalog task.

Move Up

Moves up one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Move Down

Moves down one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Properties

Displays the properties of the selected support. Because you cannot modify any properties until the support is placed, all properties on the dialog box are read- only.

Preview

Displays a picture of the selected support. The image file must be assigned to the support in the reference data.

List View

Sets the dialog box to display supports in a list view.

Grid View

Sets the dialog box to display supports in a spreadsheet-style grid view.

Check Data

Checks the consistency of the data in the grid against other data in the Catalog. This command is available only in the Catalog task.

Address

Specifies your exact location within the displayed hierarchy.

Select System Dialog Box

This dialog box displays when you select the **More...** option in the **System** list. Use this dialog box to select the system that you want. You can create new systems in the Systems and Specifications task.

Look in

Specify where you want to look for the system. Select **Workspace** to look for the system in your defined workspace only. Select **Database** to look for the system in the entire Model database.

What do you want to do?

- *Place a support by structure*

- *Modify support load* (on page 20)
- *Modify support type* (on page 20)
- *Edit support name* (on page 20)
- *Change structure connection* (on page 21)
- *Change feature connection* (on page 21)
- *Delete support* (on page 21)

Place a support by structure

1. Click **Place Support by Structure** ✕.

    TIP Select **Design** on the ribbon to design a support.

2. Select the features to support.
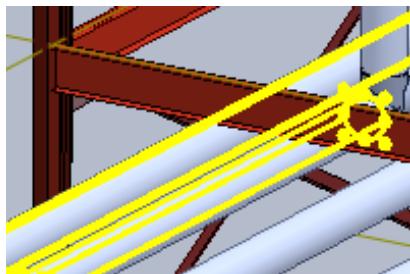
3. Click **Accept** ✓.

4. Select the supporting structure to use.
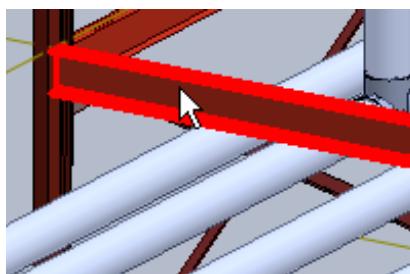
5. Click **Accept** ✓.

6. From the **Type** list, select the support type.

7. Click **Finish**.
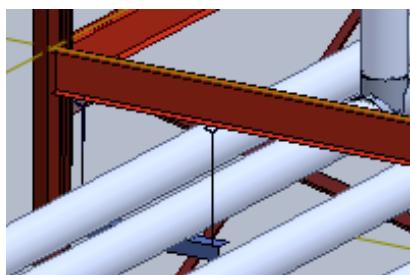
`NOTES`

- When creating a designed support, the parts you can use are available in the catalog. These parts are intelligent and contain property information that enables you to generate reports for the entire support.

- The software limits the choice of feature and structure connections to those available for the support type. This information is defined in the reference data.

- If you move or rotate a support placed by structure, the support must be in a working status. If you have appropriate permissions, you can change the status to **Working** on the **Support Properties** dialog box. See the *Configuration Tab* (on page 37) for more information.

Modify support load

1. Click **Select** on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the support or the hanger to edit.
4. Click **Edit > Properties**.
5. Type a new load value in the **Maximum Load** box.

Modify support type

1. Click **Select** on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the hanger or the support to edit.
4. On the ribbon, select a new type from the list of available types.

Edit support name

1. Click **Select** on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the hanger or support to edit.
4. Click **Edit > Properties**.
5. Select the **General** tab.
6. Type a new name for the support.

`NOTE` You can also edit the name in the **Name** box on the ribbon for a support.

Change structure connection

1. Click **Select** on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to edit.

4. Click **Toggle Connection to Structure** on the ribbon until the required connection appears.

    NOTE   The software limits the available connections based on the supporting structure and the type of support.

Change feature connection

1. Click **Select** on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to edit.

4. Click **Change Support Configuration** on the ribbon until the appropriate connection appears.

    NOTE   The software limits the available connections based on the feature and the type of support.

Delete support

1. Click **Select** on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to delete.

4. Click **Delete**.

Place Support by Point

Places a support based on the feature and the structure that you select. Use this command when the feature and the structure are parallel or when you are placing a support on a surface such as a floor or deck. You can place a single support at a location, or place multiple supports at a given distance apart.
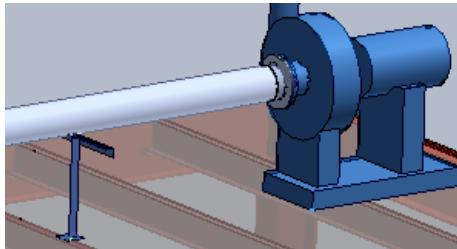
TIP You can place a support without selecting a structural reference with this command, which is useful for pipe-to-pipe supports and certain designed supports.

Support Placement by Point Ribbon

Sets options for placing a support in the model.

Properties

Opens the **Support Properties** dialog box that is used to modify support properties before placement. Edits to the default support properties before the placement are available until you change the inputs, the support type, or exit the command. This allows you to place multiple supports with the edited properties. The next time you start the command, the default support properties from the Catalog are used. For more information, see *Support Properties Dialog Box* (on page 32).

Custom Form Definition

Opens the user defined form for an object that has a form defined. It is enabled only if a proper form definition is defined for that particular object. For the remaining objects, it is disabled. For more information, see User-defined Forms.

Feature

Select the features for which you want to place a support. You must select at least one feature. Click **Reject** ✕ to clear all the selected features and start selecting the features again. Click **Accept** ✓ when you are finished selecting the features.

If you are placing the support using **Place Support by Point** , the first feature that you select is the primary feature. The system uses the cross section from the primary feature to define the support plane. If you clear the primary feature, the second feature that you select becomes the new primary feature.

Structure

Select the structure to support the feature you selected. Click **Reject** to clear all selected structures and start selecting supporting structures. Click **Accept** when you are finished selecting supporting structures.

Position

Specify the location of the support on the primary feature. This option is available only when using the **Place Support by Point** command.

Finish

Places the support using the information that you provided.

Reject

Clears all the selected objects and restarts the selection process.

Accept

Indicates that the object selection is complete.

System

Select the parent system for the support that you are placing. You can create systems in the Systems and Specifications task.

Rule

Displays the support types based on the rules defined in the reference data. When you select this option, the software uses an Assembly Selection Rule to filter the available supports in the **Type** box based on parameters such as the pipe outside diameter, distance between the supported object and the supporting object, and whether the route is above or below the supporting object. You or your project administrator have control over how the rules are defined. See the *Hangers and Supports Reference Data Guide* for more information. Clear the **Rule** option to select support types that are available.

Type

Sets the kind of support that you want to place. If you select the **Rule** option, the software lists support types based on the rules defined in the reference data. If you clear the **Rule** option, the software lists the last ten support types that you placed. Select **More** to select a different support type.

TIP When you place a designed support, the **Type** box is not available.

Change Support Configuration

Cycles through the available attachment types based on the feature and the support type. This option is not available for types that do not support multiple attachment types.

Toggle Connection to Structure

Cycles through the available attachment types based on the supporting structure and the support type. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option is not available for types that do not support multiple attachment types.

╌ **Toggle Face Position**

Cycles through the available face positions for the supporting structure. For example, you can use this option to move a U-bolt so that it does not interfere with the web of a beam. If you are placing a support from two structures, the software displays two buttons, one for each structure. This option might not be available if there are no optional face positions.

Copies

Specifies the number of copies to create along the feature length. Select the **Tools** > **Point Along** command to use this option. Type the distance between supports in the **Distance** box on the **Point Along** ribbon. The default value is 1.

NOTE This option is available only when using **Place Support by Point** .

Design

Places the support as a Design support. Select the **Design** check box, when starting the placement command, to place a Design support. Clear the **Design** check box before committing the selection of the feature, to place a Standard assembly. You can select or clear the **Design** check box any time when the Straight Feature SmartStep is active. You can even select or clear it when returning after committing the feature selection with the Green Check. If you select **Design**, the **Rule** check box is disabled.

NOTE The parts you can use to create a design support are available in the catalog. These parts are intelligent and contain property information that enables you to generate reports for the entire support.

Select Support Dialog Box

Allows selection of the type of support to be placed. This dialog box appears automatically when you select the **More...** option in the **Type** box on the ribbon. By browsing through the part hierarchy, you can find any support in the Catalog database. After you select a support, the software returns you to the model, where you can finalize placement.

 Back

Returns you to the previously selected support type or node. Use this command to navigate through the support hierarchy to the specific type you need.

 Forward

Sends you to the last selected support type or node that you moved away from by using the **Back** button. Use this command to navigate through the support hierarchy to the specific type you need.

 Up One Level

Brings up the next highest level of the support catalog hierarchy. Use this command to navigate through the support hierarchy to the specific type you need.

 Copy

Copies the selected object. This command is available only in the Catalog task.

 Paste

Pastes a copied object. This command is available only in the Catalog task.

✕ Delete

Deletes the selected object. This command is available only in the Catalog task.

↶ Undo

Reverses the most recent operation. This command is available only in the Catalog task.

New Object

Creates a new object. This command is available only in the Catalog task.

Move Up

Moves up one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Move Down

Moves down one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Properties

Displays the properties of the selected support. Because you cannot modify any properties until the support is placed, all properties on the dialog box are read- only.

Preview

Displays a picture of the selected support. The image file must be assigned to the support in the reference data.

List View

Sets the dialog box to display supports in a list view.

Grid View

Sets the dialog box to display supports in a spreadsheet-style grid view.

Check Data

Checks the consistency of the data in the grid against other data in the Catalog. This command is available only in the Catalog task.

Address

Specifies your exact location within the displayed hierarchy.

Select System Dialog Box

This dialog box displays when you select the **More...** option in the **System** list. Use this dialog box to select the system that you want. You can create new systems in the Systems and Specifications task.

Look in

> Specify where you want to look for the system. Select **Workspace** to look for the system in your defined workspace only. Select **Database** to look for the system in the entire Model database.

What do you want to do?

- *Place a support at a specific location* (on page)
- *Place multiple supports along a feature* (on page 29)
- *Modify support load* (on page 20)
- *Modify support type* (on page 20)
- *Edit support name* (on page 20)
- *Change structure connection* (on page 21)
- *Change feature connection* (on page 21)
- *Delete support* (on page 21)

Place a support at a specific location

1. Click **Place Support by Point** 🦅.

    TIP Select **Design** on the ribbon to design a support.

2. Select the features to support.
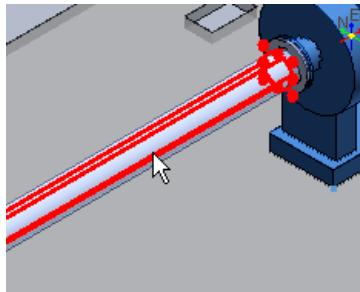
3. Click **Accept** ✓.

4. Select the structure to use to support the selected feature.

    TIP This step is optional. For example, if you are placing a support between two features only, then you do not need to select a supporting structure.
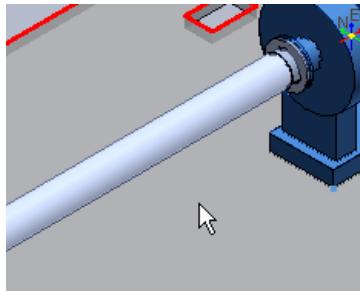
5. Click **Accept** ✓.

6. Do one of the following:

    a. If the **Rule** is checked, select the support type in the **Type** list.

    b. If the **Rule** is not selected, select **More** in the **Type** list, and then navigate the catalog browser to locate the appropriate support.

7. Select the support type.

8. Identify the location along the feature to place the support.

    TIPS

    ▪   You can use the **Tools > Point Along** command to identify this point.

- You can use SmartSketch to locate the point on the edge of a slab or wall or on the centerline of a beam to use as the graphical reference for support placement.
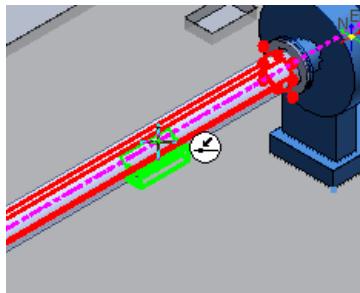
9. Click **Finish**.
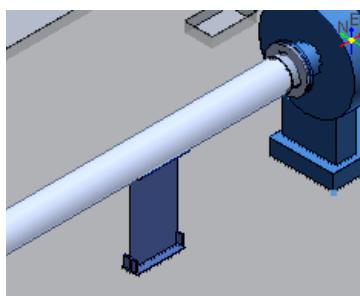
NOTES

- When creating a design support, the parts you can use are available in the catalog. These parts are intelligent and contain property information that enables you to generate reports for the entire support.

- The software limits the choice of feature and structure connections to those available for the support type. This information is defined in the reference data.

Place multiple supports along a feature

1. Click **Tools > Point Along**.

2. Click **Reference** on the ribbon, and then select the feature along which you are planning to place supports.

3. Identify the point to measure from on the feature.

4. Click **Place Support by Point** on the vertical toolbar.

5. Select the feature to support.
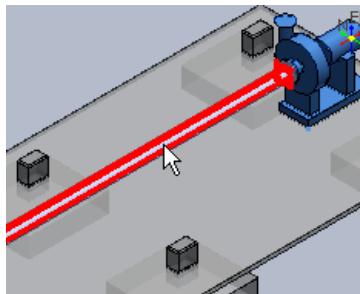
6. Click **Accept** ✓.

7. Select a structure to support the selected feature.
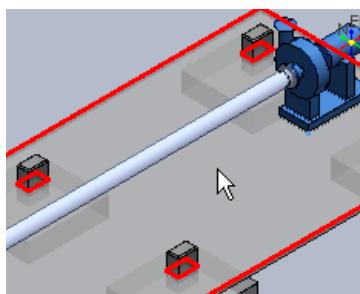
8. Click **Accept** ✓.

9. Do one of the following:

   a. If the **Rule** is checked, select the support type in the **Type** list.

   b. If the **Rule** is not selected, select **More** in the **Type** list, and then navigate the catalog browser to locate the appropriate support.

10. In the **Distance** box on the **Point Along** ribbon, specify the distance from the measure-from point that you identified in step 3 to the first support.

The software displays the first support at the distance that you defined.
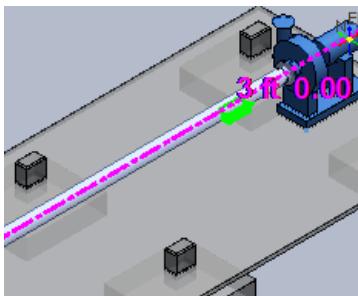
11. Click in the graphic view to position the first support.

12. In the **Copies** box, type the number of supports to place.

13. Click **Finish**.

Modify support load

1. Click **Select** ⌕ on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to edit.

4. Click **Edit > Properties**.

5. Type a new load value in the **Maximum Load** box.

Modify support type

1. Click **Select** ⌕ on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the hanger or the support to edit.

4. On the ribbon, select a new type from the list of available types.

Edit support name

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the hanger or support to edit.
4. Click **Edit > Properties**.
5. Select the **General** tab.
6. Type a new name for the support.

NOTE You can also edit the name in the **Name** box on the ribbon for a support.

Change structure connection

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the support or the hanger to edit.
4. Click **Toggle Connection to Structure** ⬚ on the ribbon until the required connection appears.

NOTE The software limits the available connections based on the supporting structure and the type of support.

Change feature connection

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the support or the hanger to edit.
4. Click **Change Support Configuration** ⬚ on the ribbon until the appropriate connection appears.

NOTE The software limits the available connections based on the feature and the type of support.

Delete support

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Support** from the **Locate Filter** list.
3. Select the support or the hanger to delete.
4. Click **Delete** ✕.

Support Properties Dialog Box

Specifies properties for the support that you have selected.

See Also

General Tab (Support Properties Dialog Box) (on page 32)
Definition Tab (on page 37)
Relationship Tab (on page 106)
Configuration Tab (on page 37)
Notes Tab (on page 39)
Drawing Tab (Support Properties Dialog Box) (on page 40)

General Tab (Support Properties Dialog Box)

Displays the support properties that you can edit or that are automatically determined by the software on placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you select more than one support and then select the **Properties** command, only the common properties among the selected supports display.

During placement, property edits that you make before placement to the default support properties are available until you change the inputs, change the support type, or exit the command. The next time you start the command, the default support properties from the Catalog are used.

When viewing properties for a single support, the following properties display. More properties may display depending on what you defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the support. Support properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, and **Responsibility**. You select the category for which to define values by using the **Category** option.

The **Restraints** category provides generic support types (for example, guide, anchor) and includes a symbolic graphic. The correct restraint information displays and can be updated.

NOTE   Support and part properties can appear in any category. The custom interfaces in the reference data control how the properties are grouped into categories.

Type

Displays the type of support. You can select a different support type if you want.

Show Dimensional Legend

Click this button to view a picture of the support with dimensions labeled.

Standard

System

Displays the parent system of the selected support.

Type Selection Rule

Displays how this support is selected. If **True**, the software selects the support. If **False**, you can select the support from the catalog.

Name

Displays the unique name of the selected support. If you type a new name, the **Name Rule** value changes to **User Defined**. Otherwise, the name is defined using the specified naming rule.

Name Rule

Defines how the support is named. If you specify the support name, this box displays **User Defined**. If the software names the support, this box displays the name of the naming rule used to name the support.

Primary Run

Displays the name of the feature that the support is supporting.

Maximum Load

Specifies the maximum load that the support must carry.

BOM Description

Specifies the Bill of Materials description of the part. The description includes the size, finish, and length where applicable. The default values are defined by the catalog and are read-only for standard supports. The values can be edited for designed supports.

TIP You can control the units and number of decimal places of the BOM using the HGrStructuralBOMUnits and HgrStructuralBOMDecimals rules in the HS_System.xls workbook on the **HgrRules** sheet. Your administrator must bulk load the workbook into Catalog using the **Add, Modify, and Delete** option if you make edits to these rules.

Support Type

Specifies the general class to which the support belongs. If you want to add, edit, or remove values that are available for selection, edit the **HngSupSupportType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Auto Face Connect

Select **Face Nearest to Route** to have the support always connect to the structure face nearest the routed object. The support also automatically updates to select the closest structure face if the route object should move to another side of the structure object. Select **Default** to have the support connect to the structure faces in the order defined in the assembly information rule.

NOTE Each support type has its own set of specific properties that can be adjusted. The following two are examples for a specific support type.

Support Direction

Overrides the direction of the support symbol shown in ISOGEN drawings. You can use the **Support Direction** property to control the support direction that is returned to the ISOGEN drawings. Support Direction is a codelist property. The following values are supported:

- **Computed** (default value) - Specifies that the support direction must be calculated by the software.

- **+X** - Specifies the global east direction.

- **-X** - Specifies the global west direction.

- **+Y** - Specifies the global north direction.

- **-Y** - Specifies the global south direction.

- **+Z** - Specifies the global up direction.

- **-Z** - Specifies the global down direction.

NOTES

- You can override the direction of a support when the support is placed without a supporting structure as the software can not calculate the support direction accurately.

- For **Move**, **Rotate Object**, or **MDR** of support along with the route, the software updates the support direction values based on the rotation of the support.

- For **Move**, **Rotate Object**, or **MDR** of route only, if the rotation of the route is more than 45 degrees, the software resets the value to **Computed**.

Reporting Requirement

Specify whether this object is reported.

Reporting Type

Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

Support Discipline

Displays the discipline of the supported object, such as piping, duct, or another discipline. If you want to add, edit, or remove values that are available for selection, edit the **HngSupSupportType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Shear Lug

Specifies that the support should have shear lugs or not.
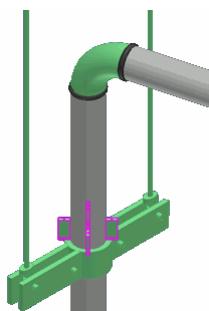
Shear Lugs

Clamp Angle

Specifies the angular offset for the support. The default angle is 0 degrees. This property is only available for supports that use the AngularRigidJoint joint type and therefore, is not available for all supports.
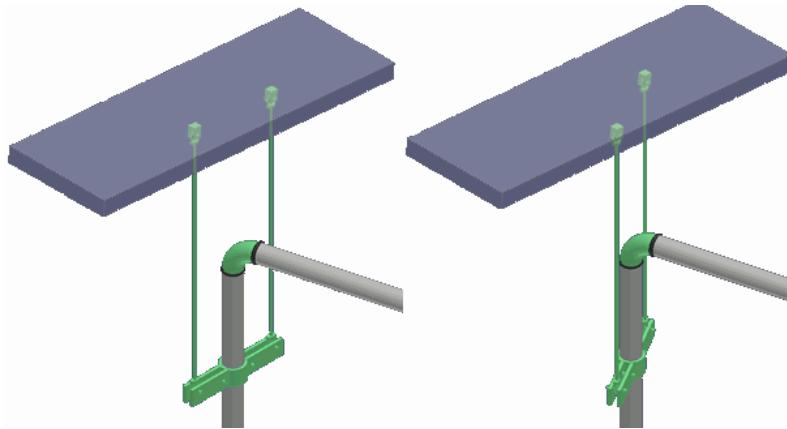
Clamp Angle set to 0 and 30 degrees

Weight and CG

Displays the center-of-gravity and the weight of the selected support. The center-of-gravity locations are displayed in global system coordinates along the X-, Y-, and Z-axes.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

Definition Tab

The **Definition** tab displays the object properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you select more than one object and then select the properties command, only the common properties between the selected objects appear.

The properties that appear depend on what you defined in the reference data. Refer to the *Hangers and Supports Reference Data Guide* for more information on the properties.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.

- For plate and profile system part properties, parent systems are shown.

- For assembly connection properties, all connected objects are shown.

- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.

- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a

requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Drawing Tab (Support Properties Dialog Box)

Displays the drawing properties for the support. Changing the view orientation allows you to control how drawing by query views display. The views associated with each package are controlled by the **IJDwgPackageOverride** bulkload sheet. For more information, see the *Drawings and Reports Reference Data Guide*.

Package

Specifies the drawing query type package associated with the support. Click **More** to display a list of available packages.

View Name

Displays the name of the view.

Orientation

Displays the orientation associated with the view. If you change the value in this box, the corresponding **Behavior** box changes to **Override**. The orientation of the view then updates in the drawing.

Behavior

Indicates whether the view orientation is overridden from the default. Set this option to **Inherit** to keep the default orientation in the drawing, regardless of the value in the **Orientation** box.

Reset to Defaults

Returns all values to their default state for the package.

S E C T I O N 5

Place Support by Reference

Places a support by selecting a reference point in the graphic view and an object to be supported. Sometimes it is necessary to place supports before the structure has been modeled. For example, you can place a support as your structure is being modeled in another package and reference to it is not available in Smart 3D.

Place Support by Reference Ribbon

Sets options for placing a support in the model.

Properties

Opens the **Support Properties** dialog box, which is used to modify support properties. For more information, see *Support Properties Dialog Box* (on page 32).

Custom Form Definition

Opens the user defined form for an object that has a form defined. It is enabled only if a proper form definition is defined for that particular object. For the remaining objects, it is disabled. For more information, see User-defined Forms.

Feature

Select the features for which you want to place a support. You must select at least one feature. Click **Reject** ✕ to clear all the selected features and start selecting the features again. Click **Accept** ✓ when you are finished selecting the features.

If you are placing the support using **Place Support by Point** , the first feature that you select is the primary feature. The system uses the cross section from the primary feature to define the support plane. If you clear the primary feature, the second feature that you select becomes the new primary feature.

Position

Specify the location of the support on the primary feature. This option is available only when using the **Place Support by Point** command.

Reference Position

Identifies where reference point has to be positioned. This reference point is used as the structure reference point, or the location of the structure. While selecting the reference point in this step, a single line (rubber band) becomes visible in the graphic view connecting the support position and the reference point.

Reference Orientation

Orients the structure reference point. It can be set up by either rotating the CS about its default Z-axis, or by rotating the CS about its Y-axis. These rotations are used to simulate skewed and/or sloping structure, respectively. You can specify the angles in the **Angle** control and lock it.

Finish

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Places the support using the information that you provided.

Length

Locks the length of the support.

Angle

Defines the angle of the **Reference Point Rotation**.

✕ **Reject**

Clears all the selected objects and restarts the selection process.

✓ **Accept**

Indicates that the object selection is complete.

System

Select the parent system for the support that you are placing. You can create systems in the Systems and Specifications task.

Rule

Displays the support types based on the rules defined in the reference data. When you select this option, the software uses an Assembly Selection Rule to filter the available supports in the **Type** box based on parameters such as the pipe outside diameter, distance between the supported object and the supporting object, and whether the route is above or below the supporting object. You or your project administrator have control over how the rules are defined. See the *Hangers and Supports Reference Data Guide* for more information. Clear the **Rule** option to select support types that are available.

Type

Sets the kind of support that you want to place. If you select the **Rule** option, the software lists support types based on the rules defined in the reference data. If you clear the **Rule** option, the software lists the last ten support types that you placed. Select **More** to select a different support type.

TIP   When you place a designed support, the **Type** box is not available.

Change Support Configuration

Cycles through the available attachment types based on the feature and the support type. This option is not available for types that do not support multiple attachment types.

What do you want to do?

- *Place support by reference* (on page)
- *Modify support type* (on page 20)
- *Change feature connection* (on page 21)
- *Delete support* (on page 21)

Place support by reference

1. Click **Place Support by Reference** 🐟.

2. Select the feature to support.
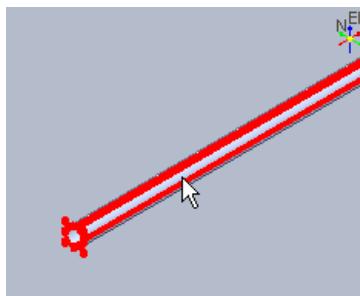
3. Click **Accept** ✓.

4. Identify the location along the feature to place the support.
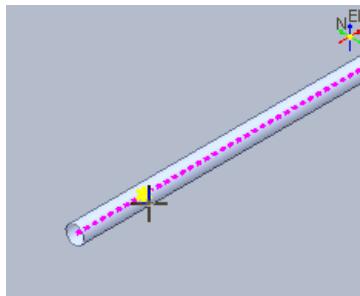
5. Do one of the following:

   - If the **Rule** check box is checked, select the support type in the **Type** list.

   - If the **Rule** check box is not checked, select **More** in the **Type** list, and then navigate the catalog browser to locate the appropriate support.

6. Select the support type.

7. Select a Reference point (in place of structure object). This reference point is used as the Structure Reference point, or the location of where the structure would be.
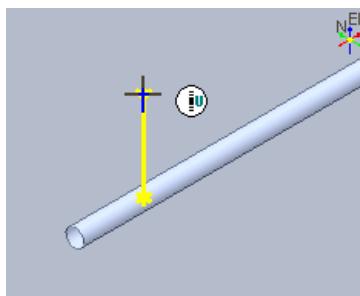
NOTE The reference point will have a default orientation, however you can also change the orientation.
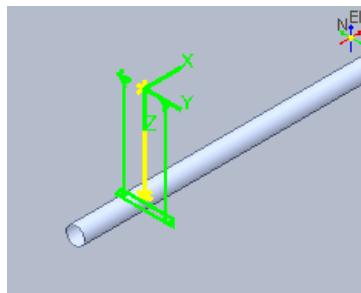
8. Click **Finish**.
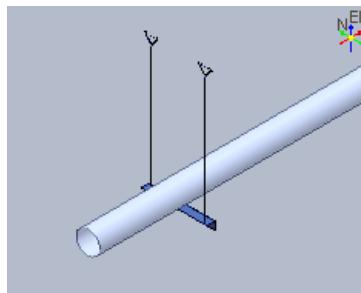
Modify support type

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the hanger or the support to edit.

4. On the ribbon, select a new type from the list of available types.

Change structure connection

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to edit.

4. Click **Toggle Connection to Structure** ⬚ on the ribbon until the required connection appears.

NOTE The software limits the available connections based on the supporting structure and the type of support.

Change feature connection

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to edit.

4. Click **Change Support Configuration** ⬚ on the ribbon until the appropriate connection appears.

NOTE   The software limits the available connections based on the feature and the type of support.

Delete support

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Support** from the **Locate Filter** list.

3. Select the support or the hanger to delete.

4. Click **Delete** ✕.

S ECTION 6

Place Part

Adds parts and welds to a support. You can also use **Place Part** to re-size or replace parts within the support.

Place Part is useful when you design a support. You can first place a standard support and then, using **Place Part**, "build it up" to customize it for your particular needs. The software checks port compatibilities and sizes as you place the parts.

If you select a standard support, click **OK** to allow the standard support to be converted to a design support. The new design support uses the same name rule as the standard support it replaces. If the name rule is user-defined, then the new design support's name is suffixed with a "*". If the standard support has a tag name rule, the same name rule will be used by the new design support.

TIP **Place Part** can locate the ports that connect support components best with certain SmartSketch locate items turned off. You should turn off all SmartSketch check marks except for **Key point** and **Reference axis aligned**. If you do not turn off all other SmartSketch options, the software locates other objects before it finds the component ports and this causes difficulties when the software builds the designed support.

Place Part Ribbon

Allows you to add or modify parts in a support.

Properties

Opens the **Properties** dialog box for the part.

Custom Form Definition

Opens the user defined form for an object that has a form defined. It is enabled only if a proper form definition is defined for that particular object. For the remaining objects, it is disabled. For more information, see User-defined Forms.

Select Part

Displays the **Select Part** dialog box. You can select a new part or part class to replace the active part.

Stretch

Extends a variable length part. This option appears only during modification (not placement). In addition, this option is available only if the active part can be extended.

Toggle Port

Toggles the port attachment of the part. This option is available only during placement (not modification).

TIP You can also press the T key to toggle the port attachment.

Name

Displays the part number from the reference data. You can edit this field.

Support Name

Displays the name of the designed support. This name also appears in the **Workspace Explorer** on the **System** tab.

Type

Displays the kind of part. After placement, this field displays the part number.

Cardinal Point

Displays the cardinal point of the active port. The cardinal point specifies the relative position of the structural section to the active port. This option is only available for rich steel. Rich steel is available by bulk loading one of the sample workbooks: HS_Str-xxxxx.xls, for example: HS_Str-CISC-8.1.xls.
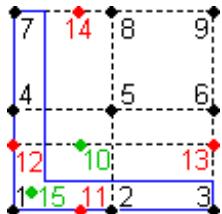
Fifteen cardinal positions are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13.

Select Part Dialog Box

Allows selection of the part to be placed. This dialog box appears when you select a support in the model. By browsing through the part hierarchy, you can find any part in the hangers and supports parts catalog. After you select a part, the software returns you to the model, where you can finalize placement.

NOTE   When creating a designed support, the parts you can use are available in the catalog. These parts are intelligent and contain property information that enable you to generate reports for the entire support.

Back

Returns you to the previously selected location. Use this command to navigate through the hierarchy to the specific part you need.

Forward

Sends you to the last selected location that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific part you need.

Up One Level

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific part you need.

Copy

Copies the selected object. This command is available only in the Catalog task.

Paste

Pastes a copied object. This command is available only in the Catalog task.

Delete

Deletes the selected object. This command is available only in the Catalog task.

Undo

Reverses the most recent operation. This command is available only in the Catalog task.

New Object

Creates a new object. This command is available only in the Catalog task.

Move Up

Moves up one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Move Down

Moves down one object. The level in the hierarchy remains the same. This command is available only in the Catalog task.

Properties

Displays the properties of the selected part. Because you cannot modify any properties until the part is placed, all properties on the dialog box are read-only.

Preview

Displays a picture of the selected support. The image file must be assigned to the part in the reference data.

List View

Sets the dialog box to display the parts in a list view.

Grid View

Sets the dialog box to display the parts in a spreadsheet-style grid view.

Check Data

Checks the consistency of the data in the grid against other data in the Catalog. This command is available only in the Catalog task.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

- *Design a support by structure* (on page 49)
- *Design a support by point* (on page 49)
- *Place a weld on a support* (on page 50)
- *Rotate a Part* (on page 51)

Design a support by structure

1. Click **Place Support by Structure** ✕ on the vertical toolbar.

2. Select the **Design** box on the ribbon.

3. Select the feature to support.

4. Click **Accept** ✓.

5. Select the supporting structure to use.

6. Click **Accept** ✓.

7. Click **Finish**.

8. Click **Place Part** ⁺📷 on the vertical toolbar.

9. Select the support that you just placed.

10. Select a support part from the catalog.

    NOTE When creating a designed support, the parts you can use are displayed in the catalog. These parts are intelligent and contain property information that enable you to generate reports for the entire support.

11. Place the part in the model.

    TIP If the connection is successful, the software flashes the support in green. If the part is placed but the ports are not connected, the software flashes the support in yellow.

12. Continue to select **Place Part** ⁺📷 until you have finished designing the support.

NOTES

- While the part is on the end of your pointer, you can use **Toggle Port** 📷 or the **T** key to toggle the port attachment.

- While the part is on the end of your pointer, you can rotate the part around a local axis at the active port. For more information, see *Rotate a Part* (on page 51).

- You can use **Stretch** to adjust the length of the part, if the part can be of variable length.

Design a support by point

1. Click **Place Support by Point** 📷 on the vertical toolbar.

2. Select the **Design** box on the ribbon.

3. Select the feature to support.

4. Click **Accept** ✓.

5. Select the structure to use to support the selected feature.

6. Click **Accept** ✓.

7. Identify the location along the feature to place the support. You can use the **Tools > Point Along** command when identifying this point.

8. Click **Finish**.

9. Click **Place Part** ⚙ on the vertical toolbar.

10. Select the support that you just placed.

11. Select a support part from the catalog.

    NOTE When creating a designed support, the parts you can use are displayed in the catalog. These parts are intelligent and contain property information that enable you to generate reports for the entire support.

12. Place the part in the model.

    TIP If the connection is successful, the software flashes the support in green. If the part is placed but the ports are not connected, the software flashes the support in yellow.

    Continue to select **Place Part** ⚙ until you have finished designing the support.

NOTES

- While the part is on the end of your pointer, you can use **Toggle Port** ⚙ or the T key to toggle the port attachment.

- While the part is on the end of your pointer, you can rotate the part around a local axis at the active port. For more information, see *Rotate a Part* (on page 51).

- You can use **Stretch** to adjust the length of the part, if the part can be of variable length.

Place a weld on a support

1. Click **Place Part** ⚙ on the vertical toolbar.

2. Select the support to add the weld to.

3. If you selected a standard support, click **OK** to acknowledge that the support will be converted to a designed support.

4. In the **Select Part** dialog box, navigate to **Parts** > **Smart Parts** > **S3D Standards** > **Miscellaneous** > **Welds** in the tree view.

5. Select the weld type to place.

6. Identify the weld's location on the support.

7. Select **Support Component** from the **Locate Filter** list.

8. Select the weld to edit, and click **Edit > Properties**.

9. Select **Welding** from the **Category** drop-down list, and define the weld properties as needed.

NOTE For information on welding category, see *Occurrence Tab (Support Component Properties Dialog Box)* (on page 53).

Place Part

Rotate a Part

You can rotate a part during initial placement using **Place Parts** or when modifying an existing part. A local coordinate system appears at the origin of the selected port on the part. The active axis of rotation is blue.

Use the arrow keys on your keyboard to rotate the part:

↑ (up arrow) - Changes the active axis in Z-Y-X order.
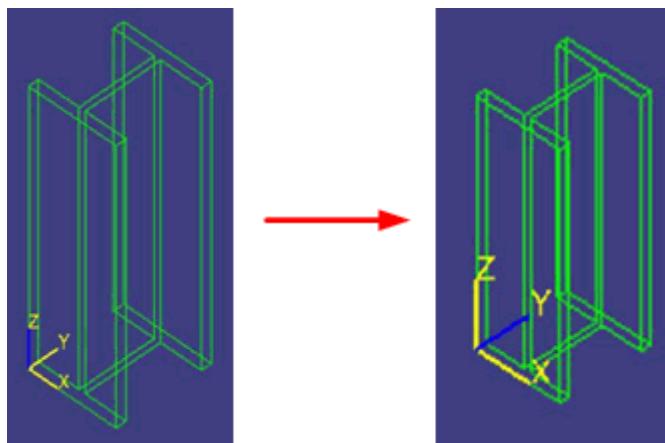
shift-↑ (shift-up arrow) - Changes the active axis in X-Y-Z order.

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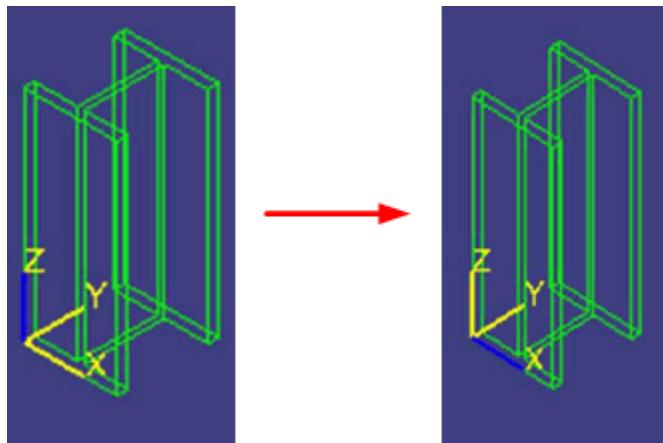
⬇ (down arrow) - Changes the active port in ascending order.
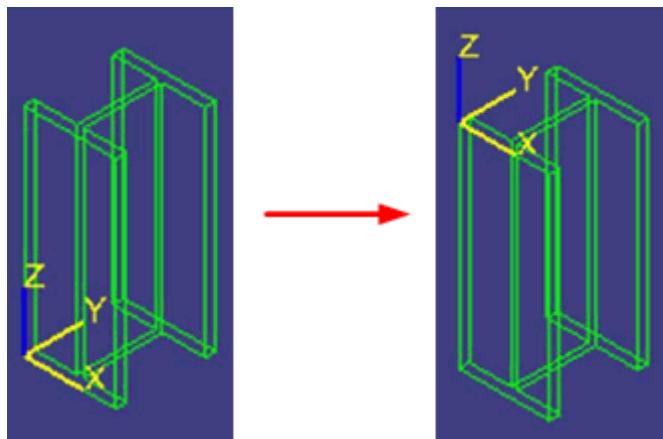
shift-⬇(shift-down arrow) - Changes the active port in descending order.
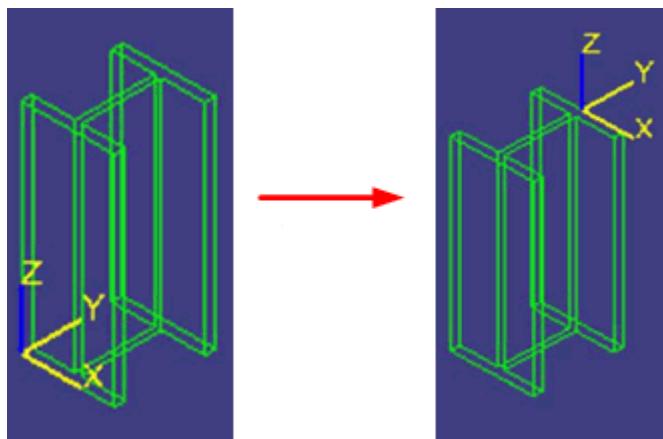
⬅ (left arrow) - Rotates the part 90⁰ counter-clockwise about the active axis.
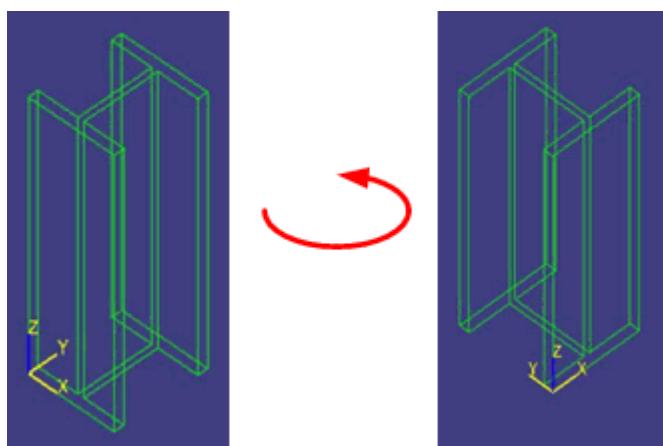
➔ (right arrow) - Rotates the part 90º clockwise about the active axis.
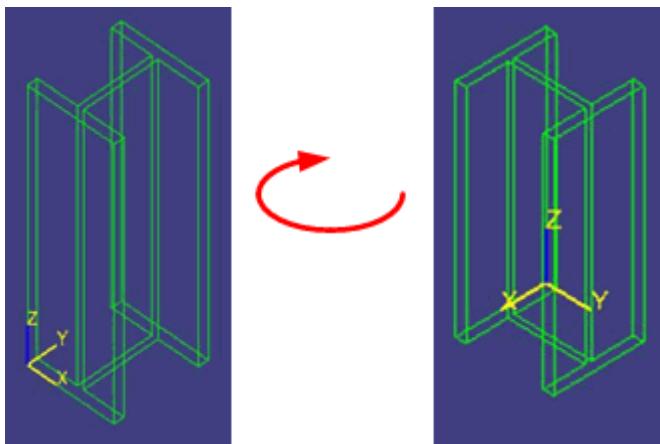
Support Component Properties Dialog Box

Sets the properties for the support parts that you have placed in the model.

See Also
Connection Tab (Support Component Properties Dialog Box) (on page 60)
Definition Tab (Support Component Properties Dialog Box) (see "*Definition Tab*" on page 60)
Occurrence Tab (Support Component Properties Dialog Box) (on page 53)
Relationship Tab (on page 106)
Configuration Tab (on page 37)
Notes Tab (on page 39)

Occurrence Tab (Support Component Properties Dialog Box)

Displays the support component properties that you can edit or that are automatically determined by the software at placement. If you select more than one support component, and then select **Properties**, only the common properties of the selected support components display.

When viewing properties for a single support component, the following properties display. More properties may display depending on what you defined in the reference data. For more information, see the Hangers and Supports Reference Data Guide.

Category

Select the properties that you want to view for the support component. Support component properties are divided into several different categories: **Standard**, **Dimensions**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, **Welding**, and **Responsibility**. You select the category for which to define values by using the **Category** option.

NOTE Support and part properties can appear in any category. The custom interfaces in the reference data control how the properties are grouped into categories.

Show Dimensional Legend

Click this button to view a picture of the component with dimensions labeled.

Standard

Name

Displays the unique name of the selected support component. If you type a new name, the **Name Rule** value changes to **User Defined**. Otherwise, the name is defined using the specified naming rule.

Name Rule

Defines how the support component was named. If you specified the support name, this box displays **User Defined**. If the software named the support, this box displays the name of the naming rule used to name the support.

Support Name

Displays the name of the parent support.

Type

Displays the type of support component. The type displayed is the part number defined in the reference data.

BOM Description

Specifies the Bill of Materials description of the part. The description includes the size, finish, and length where applicable. The default values are defined by the catalog and are read-only for standard supports. The values can be edited for designed supports.

TIP You can control the units and number of decimal places of the BOM using the HGrStructuralBOMUnits and HgrStructuralBOMDecimals rules in the HS_System.xls workbook on the **HgrRules** sheet. Your administrator must bulk load the workbook into Catalog using the **Add, Modify, and Delete** option if you make edits to these rules.

Reporting Requirement

Specify whether this object is reported.

Reporting Type

Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

Dimensions

Length

Specify the length of the selected component.

Weight and CG

Displays the center-of-gravity and the weight of the selected support component. The center-of-gravity locations are displayed in global system coordinates along the X-, Y-, and Z-axes.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating**

Color select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Welding

Specifies welding symbol parameters. Weld symbols are used by the Drawings and Reports task but welding symbol parameters are only visible on the Occurrence tab.
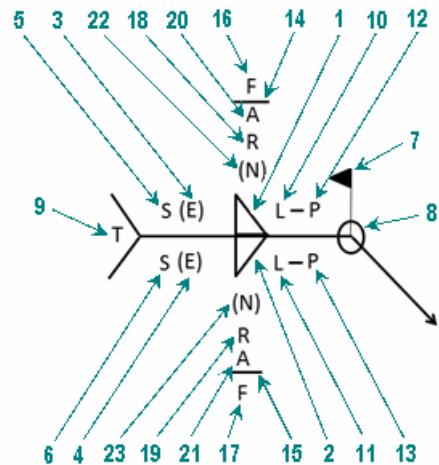
1 - Primary Side Symbol or Primary Side Groove

13 - Secondary Side Pitch or Distance Between the Ends of Adjacent Welds

2 - Secondary Side Symbol or Secondary Side Groove

14 – Primary Side Contour

3 – Primary Side Groove Size

15 – Secondary Side Contour

4 – Secondary Side Groove Size

16 – Primary Side Finish Method

5 - Primary Side Bevel Depth or Leg Length or Actual Throat Thickness or Strength

17 – Secondary Side Finish Method

6 - Secondary Side Bevel Depth or Leg Length or Actual Throat Thickness or Strength

18 – Primary Side Root Opening or Depth of Filling

7 – Field Weld

19 - Secondary Side Root Opening or Depth of Filling

8 - All Around

20 – Primary Side Groove Angle or Included Angle of Countersink

9 – Tail Notes

21 - Secondary Side Groove Angle or Included Angle of Countersink

10 – Primary Side Length

22 – Primary Side Number Of Welds

11 – Secondary Side Length

23 - Secondary Side Number Of Welds

12 – Primary Side Pitch or Distance Between the Ends of Adjacent Welds

Primary Side Symbol

For a fillet weld, specifies the fillet weld symbol for the primary side of the weld. Select **Fillet** or **None**.

Secondary Side Symbol

For a fillet weld, specifies the fillet weld symbol for the secondary side of the weld. Select **Fillet** or **None.**

Primary Side Groove

For a groove weld, specifies the weld symbol for the primary side of the weld. Select **Bevel** or **None.**

Secondary Side Groove

For a groove weld, specifies the weld symbol for the secondary side of the weld. Select **Bevel** or **None.**

Primary Side Groove Size

For a groove weld, specifies the depth of penetration of the weld on the primary side.

Secondary Side Groove Size

For a groove weld, specifies the depth of penetration of the weld on the secondary side.

Primary Side Bevel Depth or Leg Length or Actual Throat Thickness or Strength

For a groove weld, specifies the depth of bevel from the primary side.

Secondary Side Bevel Depth or Leg Length or Actual Throat Thickness or Strength

For a groove weld, specifies the depth of bevel from the secondary side.

Primary Side 2nd Leg Length or Nominal Throat Thickness

For a fillet weld, specifies the fillet throat thickness.

Secondary Side 2nd Leg Length or Nominal Throat Thickness

For a groove weld, specifies the elementary symbol on the secondary side.

Field Weld

Specifies where a weld is performed. Select **True** for a site or field weld. Select **False** for shop weld.

All Around

Specifies whether the weld extends around the perimeter of the joint. Select **True** for an all-around weld. Select **False** for a weld that is only on the sides indicated by the weld symbol.

Primary Side Supplementary Symbol

Specifies the primary side supplementary symbol. Select **None**, **Melt-Through**, **Consumable Insert**, **Permanent Backing**, **Removable Backing**, or **Spacer.**

Secondary Side Supplementary Symbol

Specifies the secondary side supplementary symbol. Select **None**, **Melt-Through**, **Consumable Insert**, **Permanent Backing**, **Removable Backing**, or **Spacer.**

Tail Notes

Specifies the text to be added in the tail notes.

Tail Note Is A Reference

Specifies whether a tail note is applied to the reference part.

Primary Side Length

Specifies the length of the weld on the arrow side.

Secondary Side Length

Specifies the length of the weld on the other side.

Primary Side Pitch or Distance Between The Ends of Adjacent Welds

Specifies the center-to-center spacing on the arrow side.

Secondary Side Pitch or Distance Between The Ends of Adjacent Welds

Specifies the center-to-center spacing distance on the other side.

Primary Side Diameter

Specifies the diameter of the spot weld on the arrow side.

Secondary Side Diameter

Specifies the diameter of the spot weld on the other side.

Primary Side Contour

Specifies the finished contour of the face of the weld.

Secondary Side Contour

Specifies the finished contour of the face of the weld.

Primary Side Finish Method

Specifies the method to use for forming the contour of the weld.

Secondary Side Finish Method

Specifies the method to use for forming the contour of the weld.

Primary Side Root Opening or Depth of Filling

Specifies the primary side root opening.

Secondary Side Root Opening or Depth of Filling

Specifies the secondary side root opening.

Primary Side Groove Angle or Included Angle of Countersink

Specifies the total included angle of the groove between parts joined by a groove weld in the primary side.

Secondary Side Groove Angle or Included Angle of Countersink

Specifies the total included angle of the groove between parts joined by a groove weld in the secondary side.

Primary Side Number Of Welds

Specifies the number of welds on the arrow side.

Secondary Side Number Of Welds

Specifies the number of welds on the other side.

Primary Side Actual Leg Length

Specifies the leg length of a fillet weld from the primary side.

Secondary Side Actual Leg Length

Specifies the leg length of a fillet weld from the secondary side.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

Definition Tab

Displays the support component properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you select more than one support component, and then select **Properties**, only the common properties among the selected support components display.

For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Connection Tab (Support Component Properties Dialog Box)

Displays the connection information for the object, the properties and their values, as defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Ports

Select the part port for which you want to view properties.

Property

Displays the name of the property as defined in the reference data.

Value

Displays the value of the corresponding property.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.

- For plate and profile system part properties, parent systems are shown.

- For assembly connection properties, all connected objects are shown.

- For the properties of a frame connection on a member, supported supporting, and auxiliary supporting parts are shown.

- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE   Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Hanger Connection Properties Dialog Box

Specifies properties for the connection that you have selected.

See Also
General Tab (Hanger Connection Properties Dialog Box) (on page 63)

General Tab (Hanger Connection Properties Dialog Box)

Displays the connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you select more than one connection and then select the **Properties** command, only the common properties among the selected connections display.

When viewing properties for a single connection, the following properties display. More properties may display depending on what you defined in the reference data. For more information, see the *Hangers and Supports Reference Data Guide* available from the **Help > Printable Guides** command in the software.

Category

Select the properties that you want to view for the connection. Connection properties are in the **Standard** category.

Standard

Connection Type

Displays the type of connection. If you want to add, edit, or remove values that are available for selection, edit the **HngSupConnectionType** sheet in the **AllCodeLists.xls** workbook in the reference data.

Connection Process

Displays the process associated with the connection. If you want to add, edit, or remove values that are available for selection, edit the **HngSupConnectionType** sheet in the **AllCodeLists.xls** workbook in the reference data.

View Hanger Ports

You can view hanger specific ports by adding the custom command using the ProgID **HgrSupAutoSupport.ShowPorts**. You can also use the **CTRL+SHIFT+P** keys to toggle the command on/off. After adding the custom command, you can trigger the command by either selecting a support or a support component only.

You can also display specific hanger ports by providing the hanger port names as arguments in the **Edit Custom Command** dialog box.

This command is specific to the Hangers and Supports environment and displays ports for the support or the support component with the following axes notations:

- x-axis - Red

- y-axis - Green

- z-axis - Blue

After adding the custom command, you can initiate the custom command from **Tools** > **Custom Commands**.

Add a Custom Command

1. Click **Tools > Custom Commands**.

2. On the **Custom Commands** dialog box, click **Add**.

3. On the **Add Custom Command** dialog box, type the ProgID you assigned to the command in Microsoft® .NET in the **Command ProgID** box.

4. Type the name you assigned to the command in the **Command name** box.

5. Type a phrase that describes the command in the **Description** box.

6. Set the **Priority** to **High**.

7. Type command line arguments in a string in the **Argument** box.

   NOTES

   - To view all the hanger ports, leave the **Argument** box empty.

   - To display specific hanger ports, type the support or support component port names as arguments in the **Argument** box.

   *The application displays the command in the **Custom Commands** dialog box.*

   NOTE Select the command from the **Command names** list to run, edit, or delete the command.

View All Hanger Ports

To view all the Hanger ports, specify the ProgID and arguments as follows:

ProgID: HgrSupAutoSupport.ShowPorts

Argument: To display all the ports of a support or a support component, leave the **Argument** box blank.

View Specific Hanger Ports

To view specific Hanger ports, specify the ProgID and arguments as follows:

ProgId: HgrSupAutoSupport.ShowPorts

Argument: Type the support or the support component port names as arguments in the **Argument** box. To display multiple ports, use a semicolon to separate the arguments.

The following example uses the port names Structure and Route to display reference ports for a Structure and a Route:
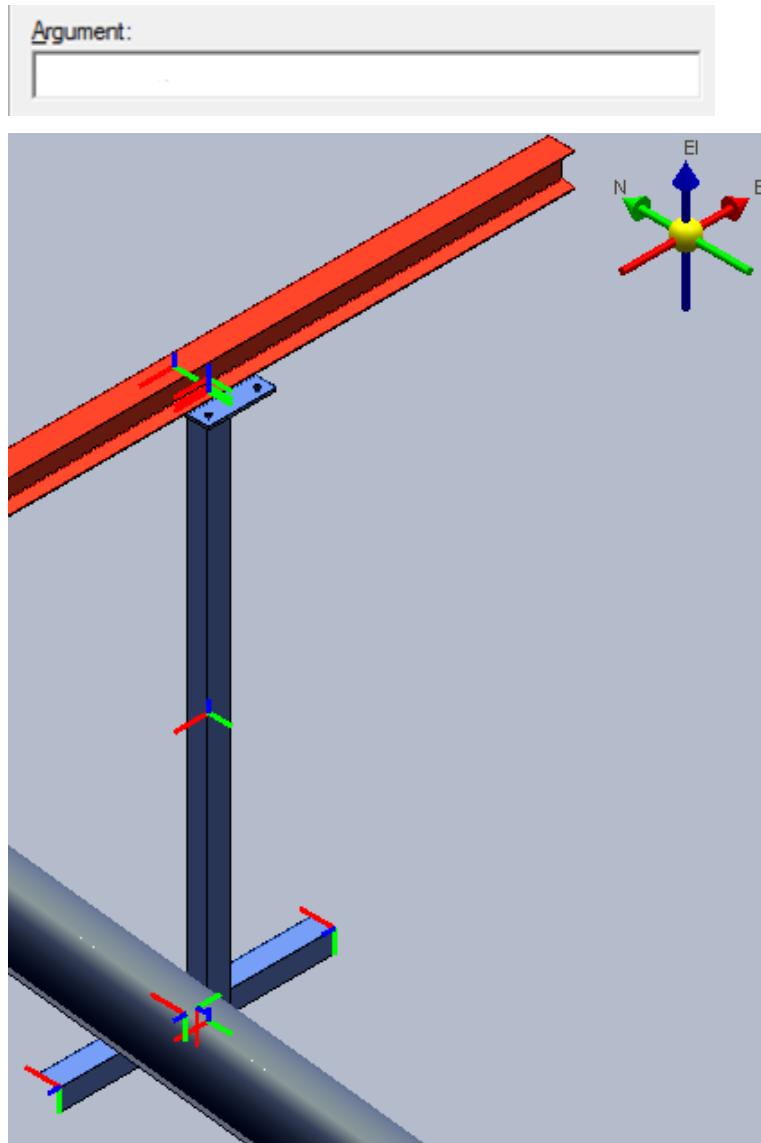
NOTE Port names are not case-sensitive.

The following example uses the port names Structure and Route to display reference ports for a Structure and a Route:
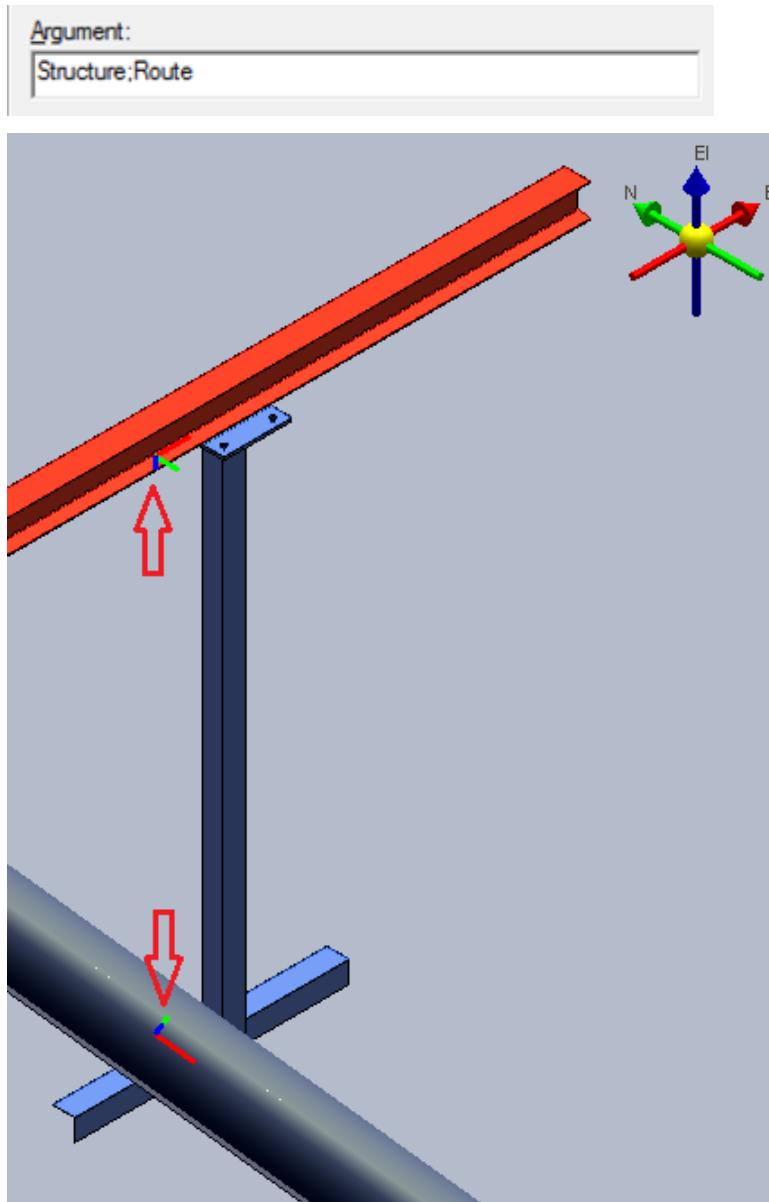
Example of valid arguments for a support are as follows:

- Route, Route_2, Route_3
- RouteAlt, RouteAlt_2
- Structure, Struct_2
- StructAlt, StructAlt_2
- World, TurnRef and so on.

Example of valid arguments for a support component are as follows:

- Pin
- Structure
- Hole
- Top
- Bottom and so on.

S ECTION 7

Drop Standard

⚡ Converts a standard support to a designed support and breaks the support into its constituent parts. You use this command to modify or delete individual parts of a standard support.

In some circumstances, you may want to modify standard supports delivered in the catalog in order to better suit the requirements of your particular support situation.

Before modifying a standard support, you must convert the support to a designed support to break the support into its constituent components. You can use the **Drop Standard** ⚡ command for this purpose.

If the support is already within a designed support and you click this command, the software breaks it into its components. If the support is not within a designed support and you click this command, then the software converts it to a designed support and breaks it into its components.

NOTES

- The software automatically creates a designed support parent and adds the standard to it when you add a part to a standard support.

- In the **Workspace Explorer**, the icon for a designed support is 🗂. The name rule of the standard support is retained by the design support after the **Drop Standard** ⚡ command is performed. The new design support uses the same tag name rule as the standard support and has a name defined as per the design support naming conventions. If the standard support name rule is user defined, the same name rule is used by the new design support and the name of the new design support is same as that of the standard support. There are few restrictions on a support hierarchy. A standard support cannot be the parent of a designed support, and a designed support cannot be the parent of another designed support.

- Associativity with the supported object is lost as a result of this command.

Drop a standard support

1. Click **Drop Standard** ⚡ on the vertical toolbar.

2. Select a standard support. The software converts the selected support to a designed support and breaks it into its parts.

NOTES

- You can select more than one standard support to convert.

- Associativity with the supported object is lost as a result of this command.

S E C T I O N   8

Add Dimensions to Supports

You can add dimensions to a support in the following ways:

- Add a note to the key points, and activate **Show dimensions** on the properties page.
- Add a control point, and activate **Show dimensions** on the properties page if key points are not available or are not at the required location.

You can also add dimensions in additional ways if reference data customization is performed by an administrator.

- Add a note to the key points available in the support, using Custom Support Definition (CSD).
- Add a dimension port to the support using the **CreateNote** function and then set the **Dimensioned** property to True in **Note** of CSD if key points are not available or are not at the required location.
- Add a control point to the support using the **CreateNote** function in CSD if the key points are not available or are not at the required location.

For more information, see "Custom Support Definition Examples" in *Hangers and Supports Reference Data Guide,* which is available using the **Help > Printable Guides** command in the software.

What do you want to do?

Add a note to key points and show dimensions interactively

To place a dimension, you need a key point or a control point for the dimension start and the dimension end. When you select **Show dimension**, a note is already added to the key point at the bottom of the vertical section using CSD. Add a note to the top of the vertical section.

1. In the **Hangers and Supports** task, click **Place Support by Structure** on the vertical toolbar.
2. Select the features to support and other needed options.
3. In the **Type** box, select **Assy_FR_IT_LS**.
4. Click **Finish.**

   The support is created.

5. In the **Workspace Explorer** or a graphic view, select the support.
6. Click **Edit** > **Properties.**

*The **Properties** dialog box appears.*

7. On the **Notes** tab, select **Keypoint4** in the **Key Point** box.

8. Click **New Note.**

9. Type the text in the **Note Text** box.

10. Select **Show Dimension**, and click **OK.**

11. Click **Tools** > **Drawing Console**

    *The **Drawing Console** window appears.*

12. Right-click the top-level model object, and select **New**.

    *The **Add Component** dialog box appears.*

13. Select **Folder** 📁, and click **OK.**

    A new folder appears in the console.

14. Right-click the folder name, and select **Rename.** Type a new name for the folder.

15. Right-click the folder, and click **New.**

    *The **Add Component** dialog box appears.*

16. Select **Volume Drawings** 📇, and click **OK**.

    *A new volume drawings component appears under the folder in the **Drawings Console**.*

17. Right-click the component, and select **Edit Template**.

18. Select a template, and click **OK.**

    The template opens in the SmartSketch Drawing Editor.

19. Click **Place Drawing View** 🖳.

20. Click in the drawing, hold down the mouse button while dragging the mouse diagonally, and release.

    A view is created on the drawing.

    *The **Drawing View Properties** dialog box appears.*

21. Type the required information on the **View** tab in the **Name**, **Description**, **Style**, **Orientation**, and **Scale** boxes.

22. Click **OK**.

23. Click **File** > **Save**, and close SmartSketch Drawing Editor.

24. In the **Space Management** task, click **Place Drawing Volume by Selection** 🎲 on the vertical toolbar.

25. Select the support in the graphic view or in the **Workspace Explorer.**

    TIP You can hold down the CTRL key to select multiple objects in the graphic view or in the **Workspace Explorer**.

26. On the **Place Drawing Volume by Selection** ribbon, select the new volume drawing template in the **Drawing Type** box.

    TIP By default, the last drawing component used in the workspace appears in the **Drawing Type** box. Click **More** to display the Select Volume Drawing Type Dialog Box.

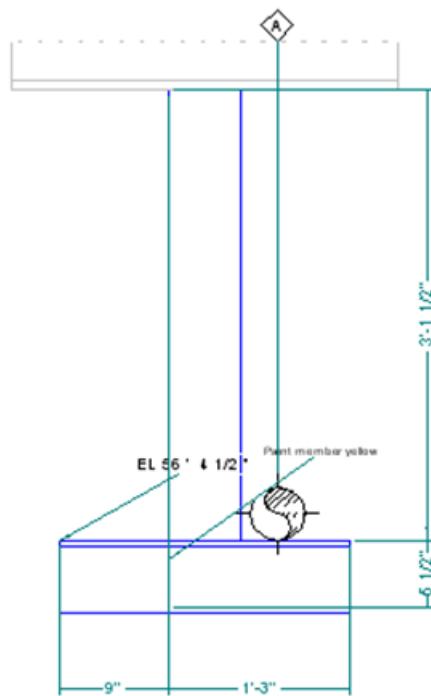27. Click **Finish** to place the volume.

28. Click **Tools** > **Drawing Console**

29. Right-click the folder containing the volume drawings component, and select **Create Drawings.**
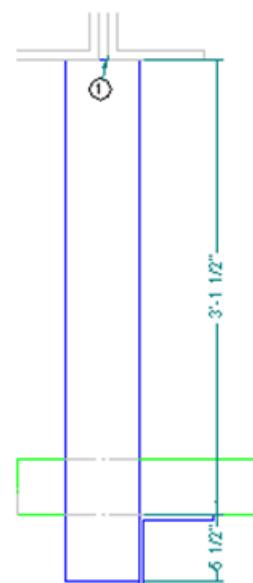
30. Open the drawing.

31. Right click the drawing view, and select **Update now**.

    The dimensions are displayed between the key points and the notes are added as shown below.

Elevation   Elevation

Add a control point and show dimensions interactively

The following procedure provides an example for using control points when key points are not readily available on a support. In this example, a control point is added to the support to obtain half length of the vertical section. To place a dimension, you need two points, a key point or a control point for dimension start, and dimension end. As a note is already added to the key point at the bottom of the vertical section using CSD, add a control point to the middle of the vertical section.

1. In the **Hangers and Supports** task, click **Place support by structure** on the vertical toolbar.

2. Select the features to support and other needed options.

3. In the **Type** box, select **Assy_FR_IT_LS.**

4. Click **Finish**.

   The support is created.

5. Click **Insert** > **Control Point.**

6. Select a location on the support where you want to add a control point. In the following example, the midpoint of the vertical section of the support is selected:
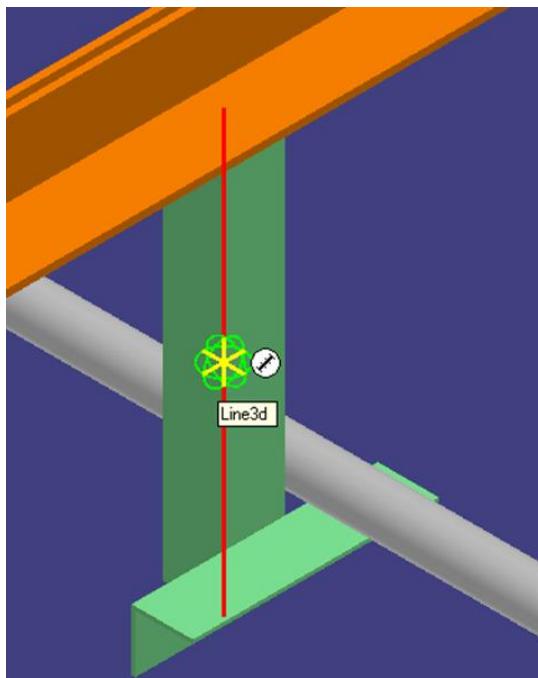
7. Click **Properties** , and modify properties as needed on the **General** tab.

8. Click **Edit > Properties.**

   *The **Properties** dialog box appears.*

9. On the **Notes** tab, select **ControlPoint4** in the **Key Point** box.

10. Click **New Note.** Select the location, purpose, and text options.

11. Select **Show dimension**, and click **OK.**
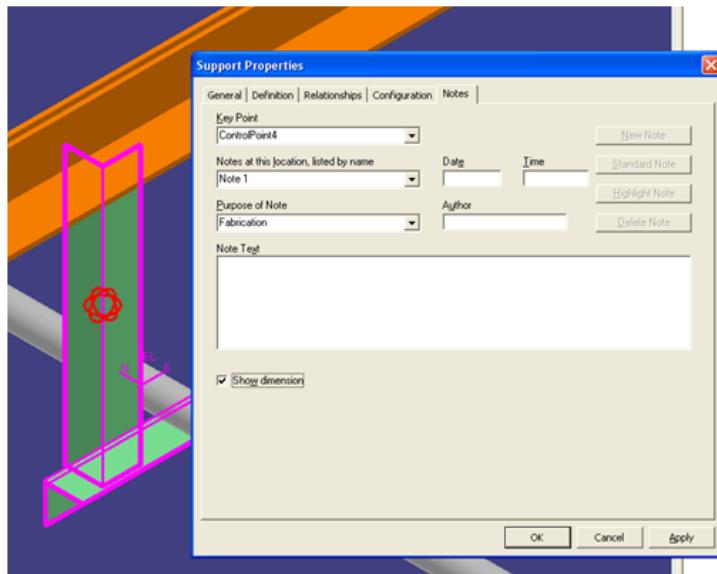
12. Click **Tools** > **Drawing Console**

    *The **Drawing Console** window appears.*

13. Create a new volume drawing component, template, drawing view, drawing volume (in the Space Management task), and drawing using the steps in *Add a note to key points and show dimensions interactively* (on page 69).

14. Open the drawing.

15. Right click the drawing view, and select **Update now**.

    The dimensions are displayed between the key point and control point as shown in the following drawing:
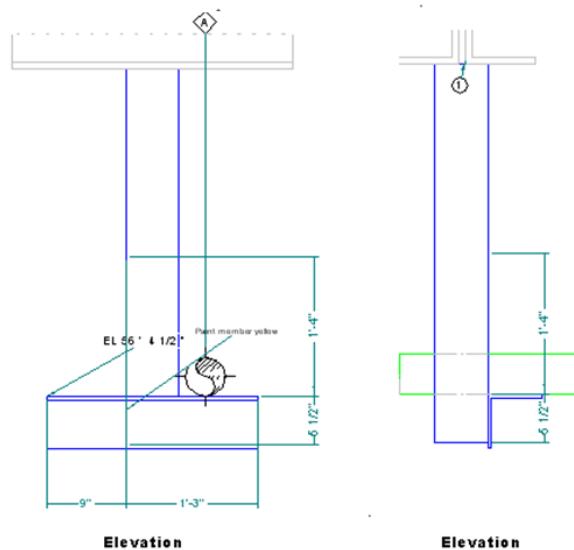
S E C T I O N 9

Start 3rd Party App

Starts a third-party application to add parts to a designed support that you have already placed in the model. Smart 3D makes information about the designed support available to the third-party application. The third-party application uses that information and returns what parts to place into the designed support and where to place them. If there are any failures, Smart 3D displays a warning message and creates an error log file, **Hangers3rdPartApp_DateandTime.log**, in the environment temp folder:

```
Part with  part number : HgrProfile_ST5x17.5 is not placed due to the excepetion : Error while constructiong the outputs
Below parts are not placed as the transaction is aborted due to the failure caused by the part :HgrProfile_ST5x17.5


!PartNumber
HgrProfile_ST3x6.25
HgrProfile_ST3x8.63
HgrProfile_ST4x9.2
HgrProfile_ST4x11.5
HgrProfile_ST5x12.7
HgrProfile_ST5x12.7
***********************************************************************************************************************
```

Define the third-party software in the HgrSup3rdPartyRule rule on the **HgrRules** sheet in the **HS_System.xls** workbook. The format of the rule is "AssemblyName, NameSpace.ClassName" where the assembly name is expected to be available on the SharedContent share defined for the Catalog. The default rule is 'Example3rdPartyPlugin, Example3rdPartyPlugin.Plugin3rdParty'.

IMPORTANT To use this command, your administrator must set the required security setting in order for the .NET assemblies to work on a network share. If you do not do this, the following error message is displayed: "Not able to get the ThirdParty Plugin from the Catalog Rule or Default Rule." Refer to [*Product Folder*]\CommonApp\SOM\Doc\3DDOTNETGuide.pdf for more information. You must install the Programming Resources to access this document.

Developing New Third-Party Plug-ins

1. Install the third-party application from which the assembly information (the part collection to be placed) is to be retrieved.

2. The third-party plug-in must be written in .NET. The .NET assembly must reference $Smart 3D\Core\Container\Bin\Assemblies\Release\HSImportAssemblies.dll from Smart 3D. A public class in the .NET assembly must be derived from the Abstract Class "Ingr.SP3D.Support.Client.HangerProg" as defined in the HSImportAssemblies.dll.

3. Retrieve all the design support information by using the "Ingr.SP3D.Support.Client.HgrParameters" class. This class object is passed as an argument of the HangerProg class functions. Pass this information to the third-party application and retrieve the assembly information (parts collection).

4. Place the new plug-in (the .NET assembly) in the symbol path and run the $Smart 3D\Core\Shared\Bin\ProgIDClassIDMapTool.exe to update the SystemSymbolConfig.xml file.

5. Update the "HgrSup3rdPartyRule" in the workbook to point to the new plug-in and bulkload the workbook into the Catalog.

6. Start this command in the **Hangers and Supports** task.

Starting Third-Party Applications from Tools

The **Start 3rd Party App** can also be started from **Tools** > **Custom Command**. This feature can run any third-party plug-ins other than those bulk loaded "HgrSup3rdPartyRule" to the Catalog. The ProgId to run this command is "HSImportAssemblies, Ingr.SP3D.Support.Client.Start3rdParty".

Use the argument for this command in the following format:

- With no string, runs plug-in from Catalog Rule.

    - "HgrSup3rdPartyRule"

- String separated by a semicolon (;). The first part is a plug-in information to run, and the second part specifies run plug-in configuration.

    - Example3rdPartyPlugin, Example3rdPartyPlugin.PlugIn3rdParty; Config

    - Example3rdPartyPlugin, Example3rdPartyPlugin.PlugIn3rdParty;

- String not separated a by semicolon (;) and contains a comma (,). The string is treated as plug-in information that runs the plug-in.

    - Example3rdPartyPlugin, Example3rdPartyPlugin.PlugIn3rdParty

- String not separated by a semicolon (;) and does not contain a comma (,). This string indicates run plug-in configuration.

    - Config (Plug-in Configuration for Plug-in in "HgrSup3rdPartyRule" Rule)

Example Third Party Plug-in
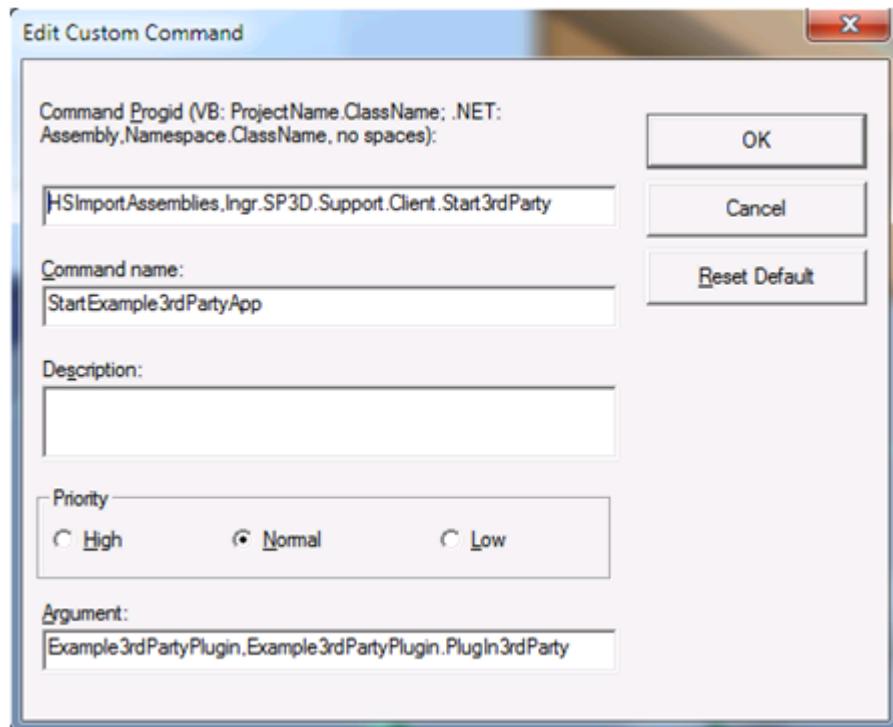
Licad Plug-in

For more information about the Licad plug-in, see
http://www.lisega.de/downloads/licad-for-smartplant-3d-2.html
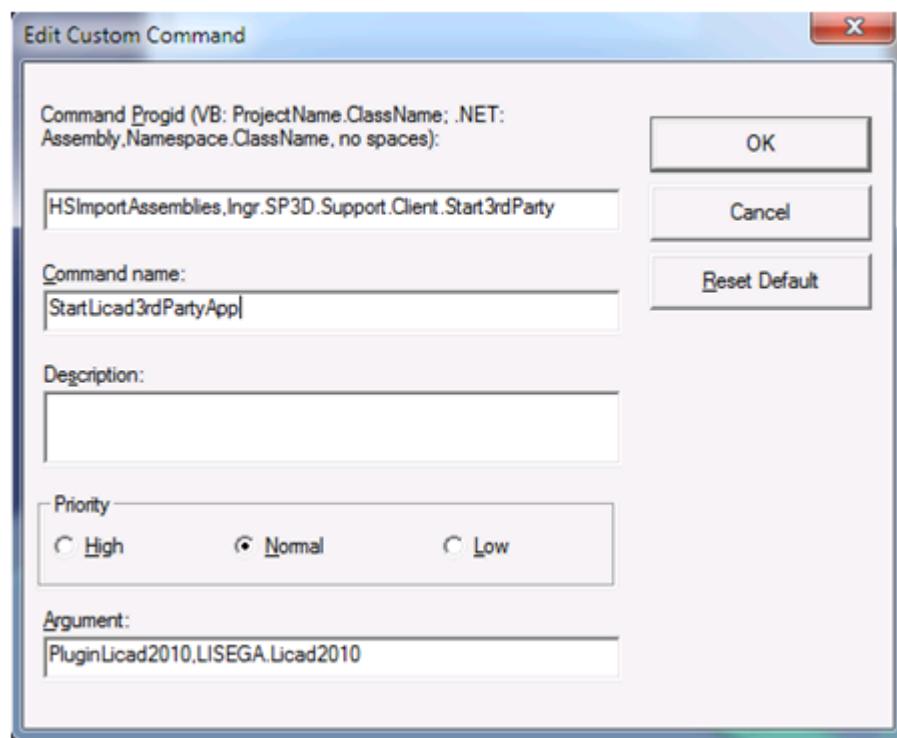
Place Linear Member Systems

Places a linear member in the model. You can place beams, columns, braces, truss elements, or cable member types using this command. You can place standard members (rolled shapes) or designed members (built-up shapes). For additional information, see Members. Use this command when you want to place members by specifying the exact start and end points.
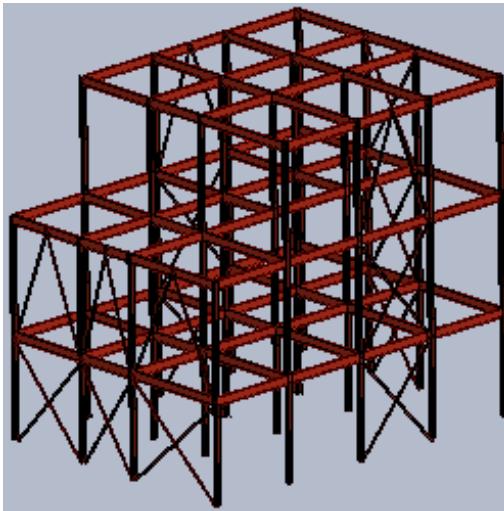

NOTES

- You can define custom member types by editing the **Structural Member Type** select list in the Catalog task.

- When a linear member system is connected directly to another member, a frame connection is created. If detailed trimming is needed in a plant database, use the **Trim Member** or **Place Assembly Connection** . If detailed trimming of the members is needed in a marine database, use **Place Member Generic Assembly Connection** in the Molded Forms or Structural Detailing task.

- When a linear member system is connected to a plate system or edge reinforcement system, the member is trimmed to the plate and a generic assembly connection is created. In marine mode, the bounding ports can be modified with **Place Member Generic Assembly Connection** in the Molded Forms or Structural Detailing task. For more information, see Place Member Generic Assembly Connection.

Place Linear Member System Ribbon

Specifies the properties for the member that you are placing. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon*.

Member Properties

Activates the *Member System Prismatic Properties Dialog Box* (on page 104). You can use this dialog box to specify additional member properties that you cannot set on the ribbon, such as material, material grade, and end releases. Because changes made on the ribbon, such as to the **Type Category**, can affect or reset properties in the dialog box, we recommend that you set properties in this dialog box after you have made your selections from the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see Members.

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see Members.

Enter AMP

Activates the *Advanced Member Positioning Ribbon*, which you can use when you want to place a member at the intersection of three objects and maintain the relationship to those objects. For example, a column that needs to stay over two walls, or bulkheads, on a lower level. You must select the **Align-Default** or **Align-Lapped** frame connections from the **Connection** box to use the advanced member positioning options.

Finish

Click to place the member in the model. **Finish** is active only when **Finish Mode** is selected.

Finish Mode

Specify whether or not **Finish** must be selected to place a member in the model. If the **Finish Mode** is selected, the software places the member in tentative mode after you identify the second end point. This tentative mode allows you to modify placement settings such as the offset, cardinal point, or frame connection properties before you commit the member to the model. If the **Finish Mode** is not selected, then the software automatically places the member in the model after you identify the second end point.

Connection

Select the frame connection type to use for the member that you are placing. If you select **By Rule**, the software determines the frame connection to use based on the geometry between the member that you are placing and existing members in the model. If you select **More**, all available frame connections display from which you can select the frame connection to use. For more information about frame connections, see Frame Connections. This option is not available if you are editing an existing member. You can change the frame connection later by editing the frame connection directly.

Connection Properties

Activates the **Connection Properties** dialog box, which is used to specify properties for the active frame connection. The properties that appear in this dialog box are described under the Connection Properties section.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* in *Place Linear Member Systems* (on page 77).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.
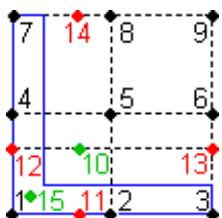
Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.

Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set using the **Coordinate System** control in **PinPoint** . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Connection Properties
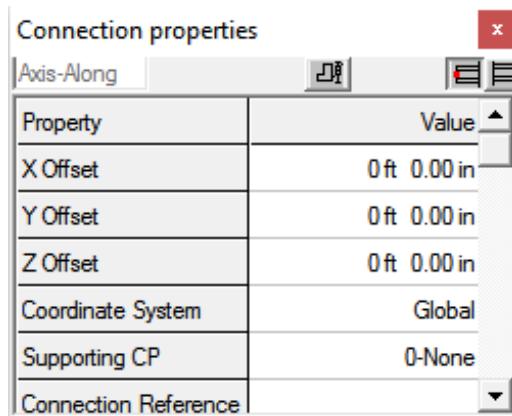
The **Connection Properties** appear only when you have selected the **Connection Properties** option. Connection properties change depending on the frame connection specified in the **Connection** option. To see the frame connection properties for the start of the member, select . To see the frame connection properties for the end of the member, select . Click to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.

TIP The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.

Seated, Flush, and Centerline Frame Connection Properties

Side

Select the side of the supporting member on which you want to place the supported member.

Offset

Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

Edge

Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. For example, when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.

Axis Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset

Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset

Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System

Specifies the coordinate system to use for the offset values. Local is the local coordinate system of the supporting member.

Supporting CP

Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Surface Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction.

Y Offset

Specifies an offset to apply in the y-direction.

Z Offset

Specifies an offset to apply in the z-direction.

Coordinate System

Specifies the coordinate system to use for the offset values.

Vertical Corner Brace Frame Connection Properties

X Offset

Specifies the offset to apply in the x-direction.

Y Offset

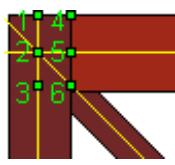Specifies the offset to apply in the y-direction.

Z Offset

Specifies the offset to apply in the z-direction.

Coordinate System

Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point

Specifies the work point location. There are six work point locations that you can select.

1 - Primary Center - Secondary Far Side
2 - Primary Center - Secondary Center
3 - Primary Center - Secondary Near Side
4 - Primary Near Side - Secondary Far Side
5 - Primary Near Side - Secondary Center
6 - Primary Near Side - Secondary Near Side

Modify Linear Member Part Ribbon

Displays the member part properties that you are editing when you select one or more linear member parts.

Member Properties

Activates the *Member Part Prismatic Properties Dialog Box* (on page 109). You can use this dialog box to specify additional member part properties, such as material, material grade, and end releases, which you cannot set on the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see Members.

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see Members.

Convert

Translates a member part to have a single stand-alone member system. This option is only available when you select a member part that belongs to a member system that has been

split.

When you place a member system, that member system has a single member part associated with it (for more information, see Members). Using the Place Splits, you can split that single member part into multiple member parts that are each still associated with the original single member system.

If you want to delete or modify only one of the member parts, you must delete the member system and all the other member parts. Deleting a member part deletes its parent member system which causes the sibling member parts to be deleted. However, using the **Convert** option you can cause a member part to have its own, new parent member system. You can then delete or otherwise modify that member part without affecting the other member parts of the original member system.

The software automatically translates the relevant split connections into frame connections. Permission groups of the original member system are used for the new member system.

System

Select the system to which the member belongs. You can define new systems in the Systems and Specifications task. Select **More** to display all systems defined in the workspace or the model. For more information, see *Select System Dialog Box* in *Place Linear Member Systems* (on page 77).

Type Category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.
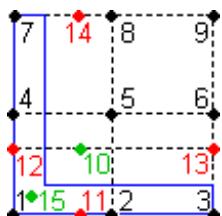
Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.
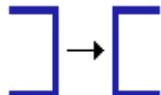
Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The

local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle, in degrees or radians, by which the cross-section is rotated about the member axis. The zero-degree position is either the Z-axis or the X-axis of the active coordinate system depending on the member orientation. The active coordinate system is set

using the **Coordinate System** control in **PinPoint** . The **Angle** property is not used when placing members using the **Seated**, **Flush**, or **Centerline** frame connections.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Advanced Member Positioning Ribbon

Specifies the properties for the member that you are placing using the advanced member positioning option. When editing a member part, this ribbon changes. For more information on the properties that are available when you are modifying a member part, see *Modify Linear Member Part Ribbon* in *Place Linear Member Systems* (on page 77).

Member Properties

Activates the *Member System Prismatic Properties Dialog Box* (on page 104). You can use this dialog box to specify additional member properties that you cannot set on the ribbon, such as material, material grade, and end releases. Because changes made on the ribbon, such as to the **Type Category**, can affect or reset properties in the dialog box, we recommend that you set properties in this dialog box after you have made your selections from the ribbon.

Start

Specify the start location of the member. After placing the first member, click **Start** to select the discrete placement method. For more information about discrete placement, see Members.

End

Specify the end location of the member. After placing the first member, click **End** to select the contiguous placement method. For more information about contiguous placement, see Members.

Exit AMP

Exits the advanced member positioning ribbon, and returns you to the standard member controls.

Connection Properties

Activates the **Connection Properties** dialog box, which is used to specify properties for the active frame connection. The properties that appear in this dialog box are described below in the *Connection Properties* section.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.
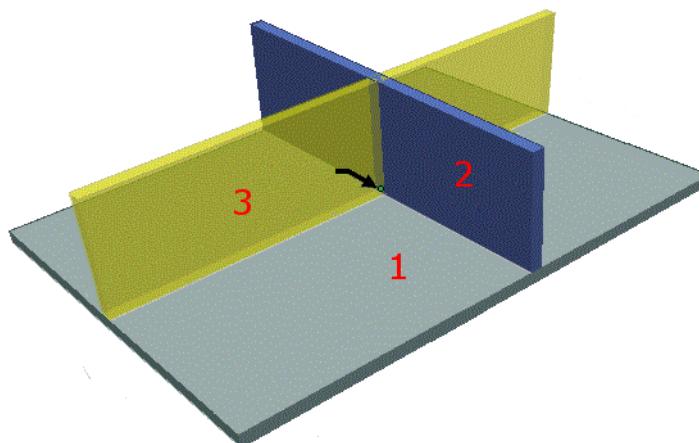
1

Specifies the bounding object.

2

Specifies the align object.

3

Specifies the intersect object.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.

- 1 - Yellow background when an input is selected.

- `1` - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

`NOTE` Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:
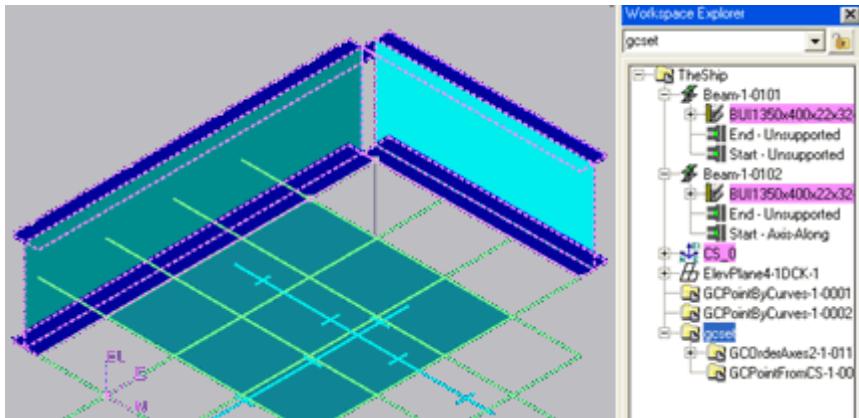
When changing an existing geometric construction to a different definition, colors represent the different value changes:

- `1` and `0.00 m` ▼ - No background color when you keep the value of the original geometric construction.

- `1` and `0.10 m` ▼ - Yellow background when you change a value.

- `1` and `0.10 m` ▼ - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

`IMPORTANT` The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Bound Offset

Type an offset distance for the point from the bound surface. This offset is in addition to the ratio you defined.

Bound Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Entering a ratio of 0.5 centers the point between the two surfaces.
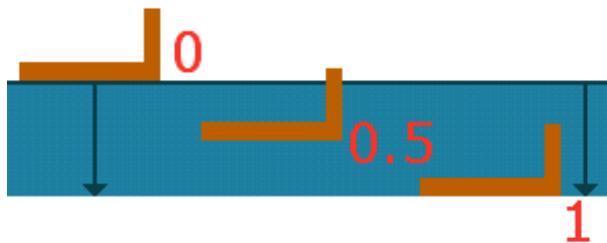
Bound Orient

Specifies how to interpret the bounding object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Align Offset

Type an offset distance for the point from the align surface. This offset is in addition to the ratio you defined.

Align Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Entering a ratio of 0.5 centers the point between the two surfaces.

Align Orient

Specifies how to interpret the align object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Intersect Offset

Type an offset distance for the point from the intersect surface. This offset is in addition to the ratio you defined.

Intersect Ratio

Type the ratio of the plate's thickness to the point position. Type **0** to place the point on the molded-form surface. Type **1** to place the point on the anti-molded-form surface. Typing a ratio of **0.5** centers the point between the two surfaces.

Intersect Orient

Specifies how to interpret the intersect object for advanced member positioning. Select **Primary** to evaluate along the primary axis. Select **Secondary** to evaluate along the secondary axis. See **Orient Options** below.

Orient Options

Specifies if the profile is interpreted along its primary orientation or its secondary orientation. If along the primary orientation, then 0.0 is the red surface and 1.0 is the black surface. If along the secondary orientation, then 0.0 is the green surface and 1.0 is the blue surface.

Tee

Includes W, S, and M types and fabricated Ts
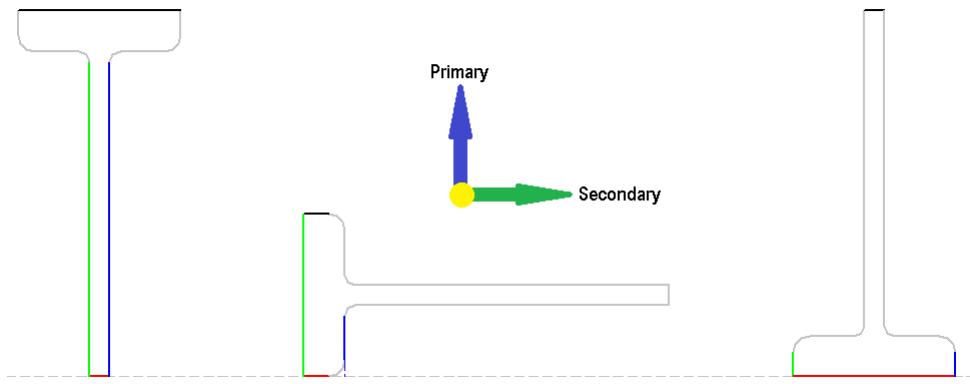
I-beams

Includes W and S types.

Angle

Includes equal and unequal leg angles.

Fabricated Angle

Flat Bar

Channel
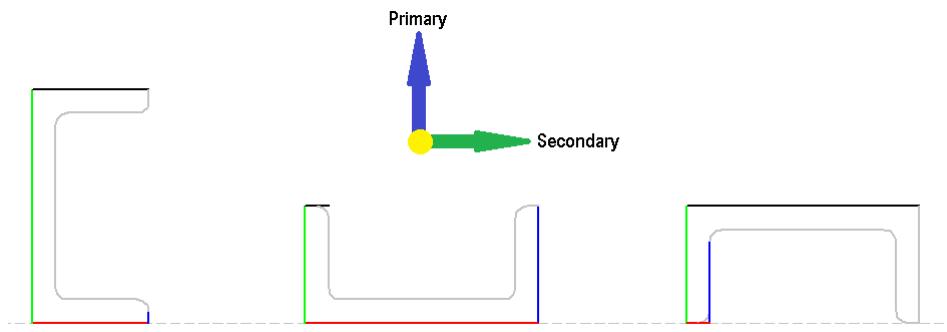
Bulb
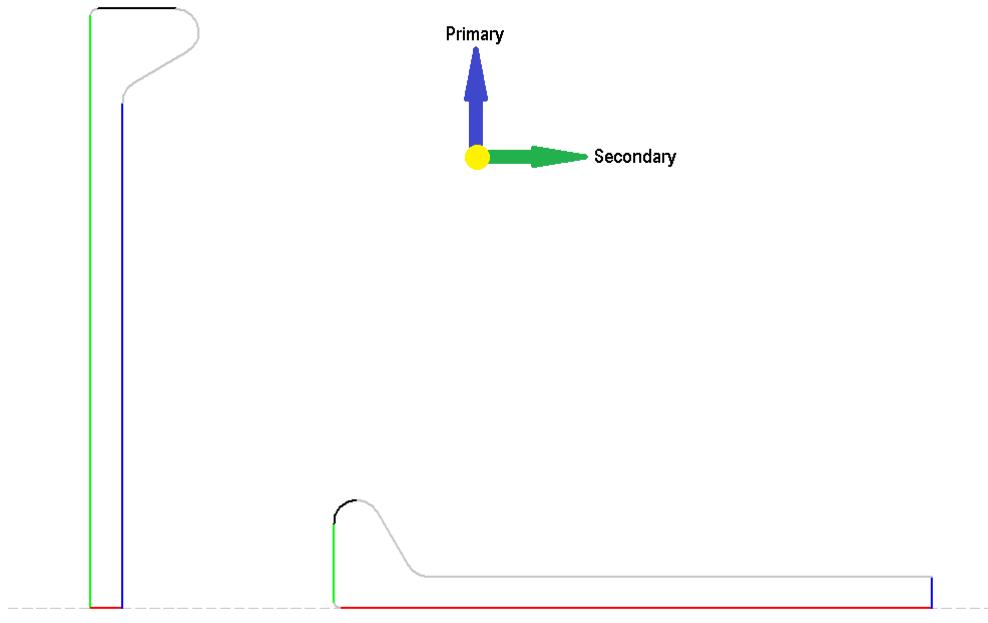
Rectangular Tube
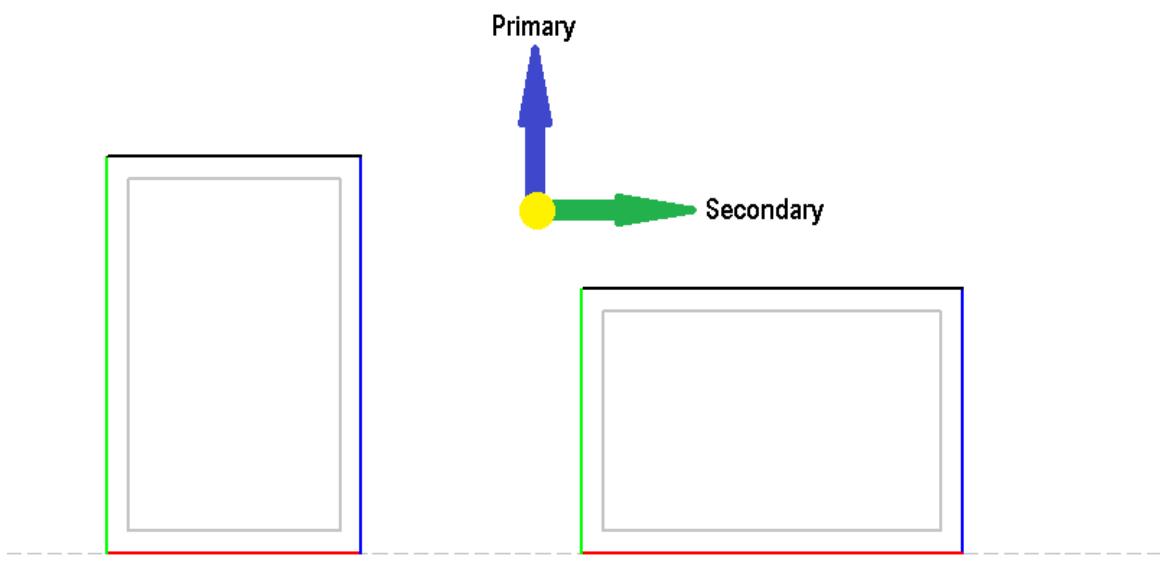
Half Round and Round Bar
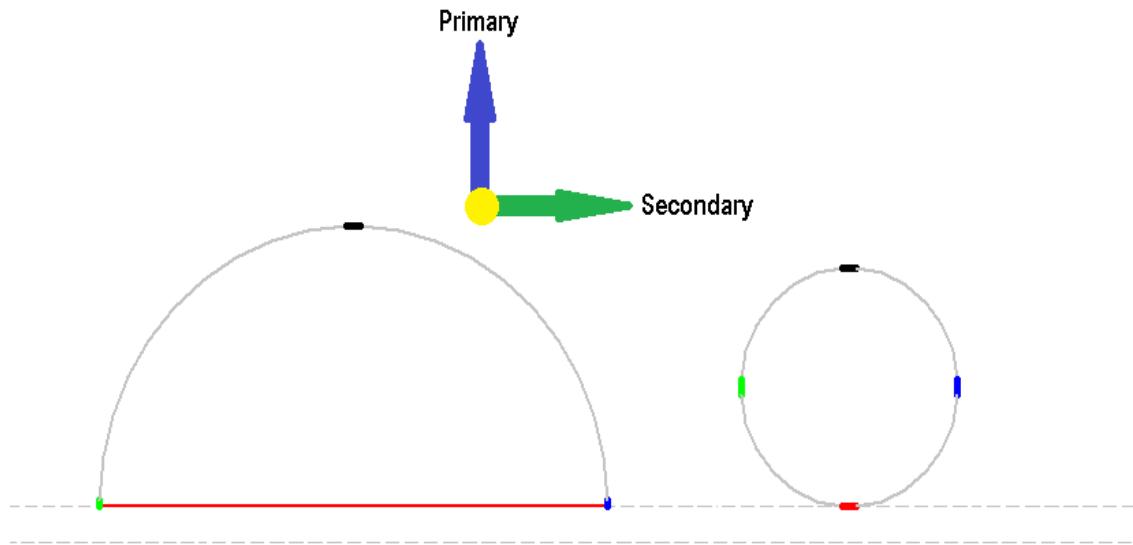
Round Tube (Pipe)

NOTE Round Tube has no faces defined along the secondary orientation.
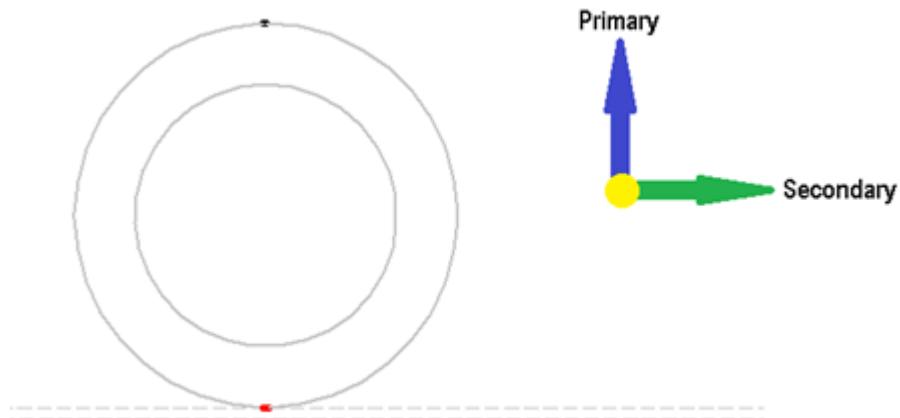
Connection Properties

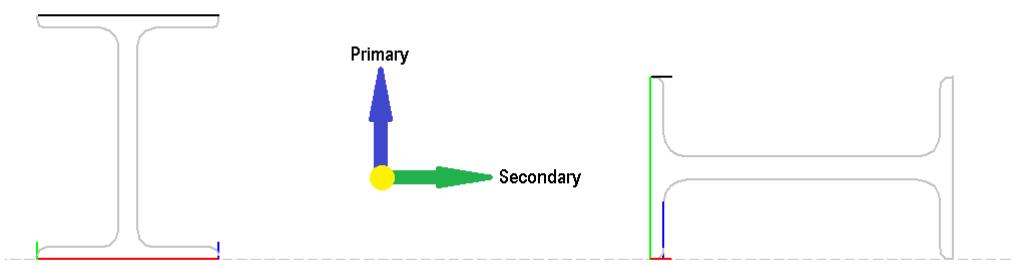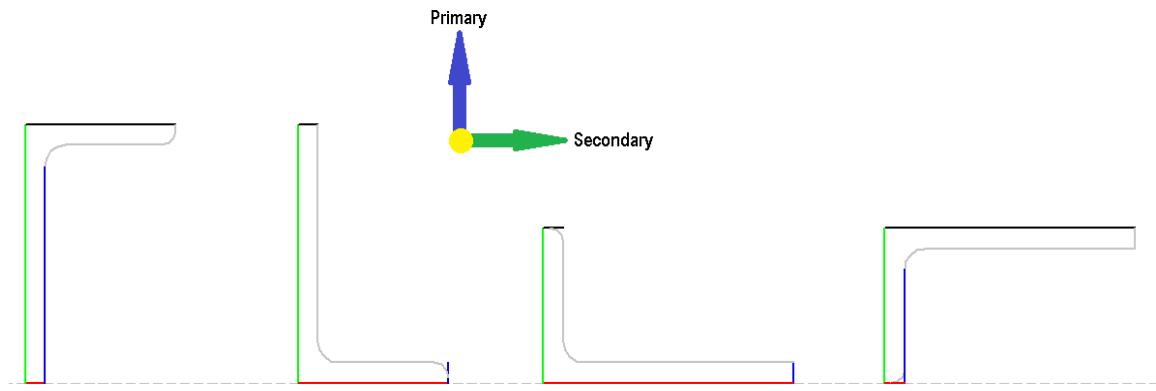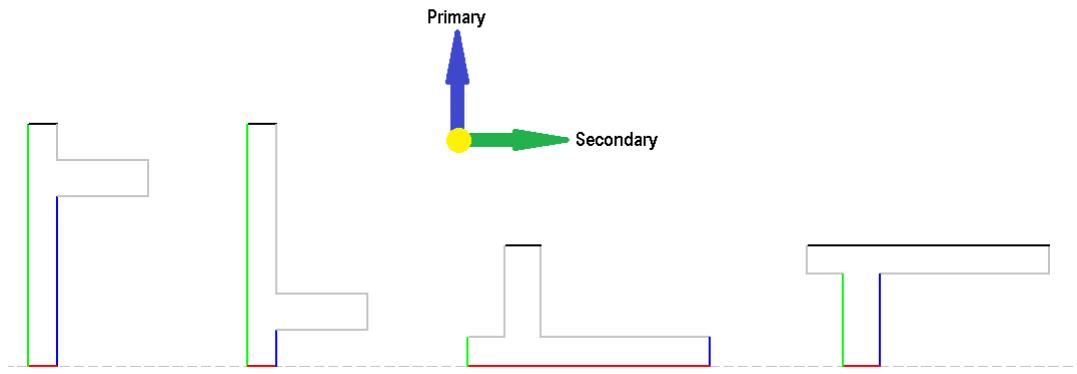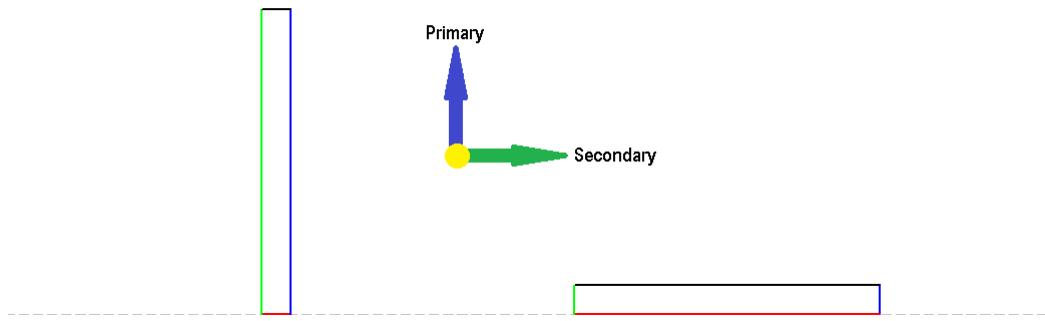The **Connection Properties** appear only when you have selected the **Connection Properties** ▣ option. Connection properties change depending on the frame connection specified in the **Connection** option. To see the frame connection properties for the start of the member, select ▤ . To see the frame connection properties for the end of the member, select ▤. Click ◫ to see a preview of the frame connection. The frame connection type appears in the upper left corner of the dialog box.

TIP The supported member is the member that you are placing. The supporting member is the existing member in the model to which you are connecting.

Align-Default/Lapped Frame Connection Properties

Side Justification

Select the side of the plane on which to place the member.

- **None** - Aligns the member axis so that it is placed on the surface.

- **Seated** - Sets the member on top of the surface. The selected **Orientation Side** is nearest the surface.

- **Center** - The center of the cross-section is placed on the surface. The center is calculated by the top-bottom, left-right bounds of the cross-section shape.

- **Flush** - Uses the alignment surface's top and bottom extent to position the supported member. The supported member typically lies within the body of the alignment plane with one edge of the member flush with the alignment surface but can be offset.

Orientation Control

If **On**, the member angle is controlled by the **Orientation Angle** value defined for the Align frame connection. If **Off**, you can set the angle of the member independently of the align surface by using the **Rotation** property for the member part.

Orientation Side

Specifies the side of the supported member's cross-section that is mated to the alignment surface. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

Orientation Angle

When **Orientation Control** is **On**, this angle is added to the selected **Orientation Side** to rotate the member. If the Align frame connection is used on both ends of the member and you define a different angle value for each end, the angle defined at the start end of the member is used.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option is when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.

Extend Distance

Specifies the offset distance from the bounding surface along the member's axis.

Side Clearance

Specify the offset distance perpendicular to the align surface to place the member axis.

Lateral Distance

Specifies the offset distance in the plane of the align surface to place the member axis.

Select System Dialog Box

This dialog box displays when you select the **More...** option in the **System** list. Use this dialog box to select the system that you want. You can create new systems in the Systems and Specifications task.
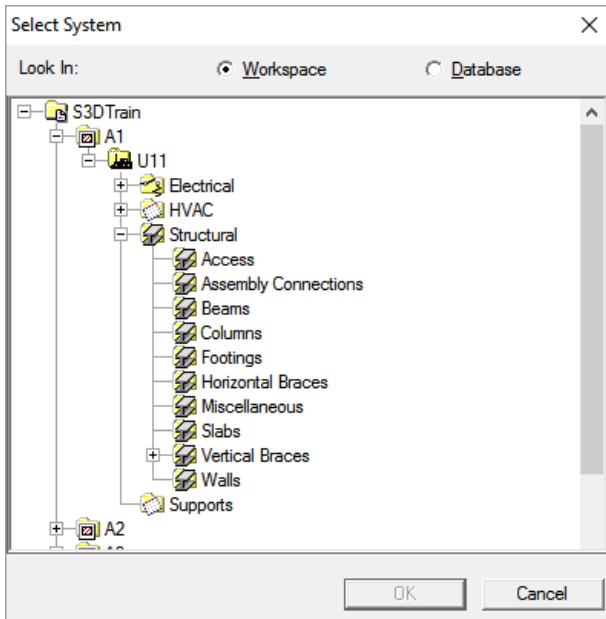
Look in

> Specify where you want to look for the system. Select **Workspace** to look for the system in your defined workspace only. Select **Database** to look for the system in the entire Model database.

Select Section Dialog Box

Allows selection of the type of section to be placed. This dialog box appears when you click the **More** option on the **Section Name** list. By browsing through the hierarchy, you can find any section in the Catalog database. After you select a section, the software returns you to the model, where you can finalize placement.
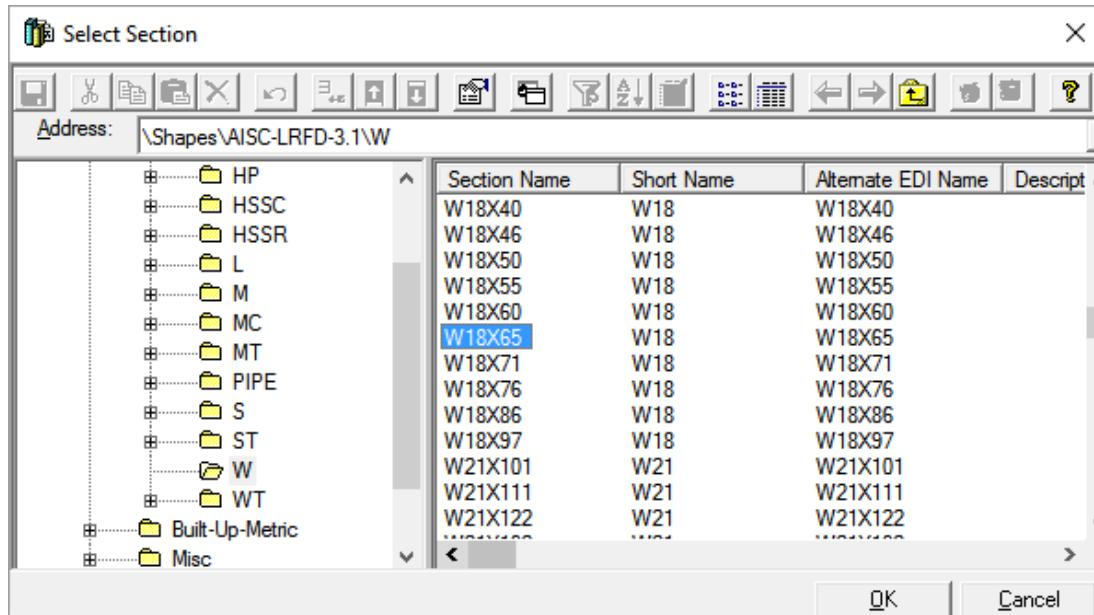
Properties

Displays the properties of the selected section. Because you cannot modify any properties until the section is placed, all properties on the dialog box are read-only.

Preview

Displays a picture of the selected section. The image file must be assigned to the section in the reference data.

Filter

Allows you to filter catalog data to help find the subset of data that you want to work with, similar to Microsoft Excel.

Sort

Sorts the catalog data by column to help you find like items.

Customize Current View

Defines with columns in the data you want to see.

List View

Sets the dialog box to display sections in a list view.

Grid View

Sets the dialog box to display sections in a spreadsheet-style grid view.

← Back

Returns you to the previously selected section type or node. Use this command to navigate through the hierarchy to the specific type that you need.

⇨ Forward

Sends you to the last selected section type or node that you moved away from by using the **Back** button. Use this command to navigate through the hierarchy to the specific type that you need.

⬆ Up One Level

Brings up the next highest level of the catalog hierarchy. Use this command to navigate through the hierarchy to the specific type that you need.

Address

Specifies your exact location within the displayed hierarchy.

What do you want to do?

- *Place members using discrete placement* (on page 97)
- *Place members using contiguous placement* (on page 98)
- *Place a member using finish mode* (on page 99)
- *Edit member system properties* (on page 99)
- *Edit member part properties* (on page 100)
- *Edit a frame connection* (on page 100)
- *Delete a member system* (on page 100)
- *Convert a member part* (on page 100)
- *Modify the cardinal point of a member* (on page 101)
- *Modify the end releases of a member* (on page 101)
- *Modify the angle of a member* (on page 101)
- *Modify the cross-section of a member* (on page 102)
- *Modify the material of a member* (on page 102)
- *Modify the material grade of a member* (on page 102)
- *Modify the type of member* (on page 102)
- *Move a member* (on page 103)
- *Move one end of a member* (on page 103)
- Move one end of multiple members

Place members using discrete placement

1. Click **Place Linear Member System** on the vertical toolbar.

2. In the **Connection** box, select a frame connection type.

    TIP If you are unsure of which frame connection type to use, review Frame Connections. You can also select the **By Rule** option to allow the software to select automatically a frame connection type.

3. In the **Type category** box, select the member type category to place.

4. In the **Type** box, select the member type to place.

5. Specify the start location, or first point, of the member.

    TIP You can use the frame connection of another member as the start or end location of the member that you are placing.

6. Specify the end location, or second point, of the member.

7. Click **Start** to specify the start location of the next member.

8. Specify the start location of the next member.
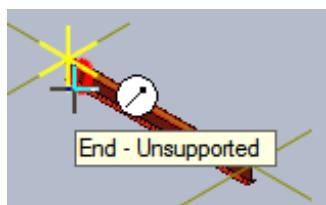
NOTES

-   To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** ⊗ option is selected.

-   You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** ⬩ and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Place members using contiguous placement

1. Click **Place Linear Member System** 📦 on the vertical toolbar.

2. In the **Connection** box, select a frame connection type.

    TIP If you are unsure of which frame connection type to use, review Frame Connections. You can also select the **By Rule** option to allow the software automatically to select a frame connection type.

3. In the **Type category** box, select the member type category to place.

4. In the **Type** box, select the member type to place.

5. Specify the start location, or first point, of the member.

    TIP You can use the frame connection of another member as the start or end location of the member that you are placing.

6. Specify the end location, or second point, of the member.
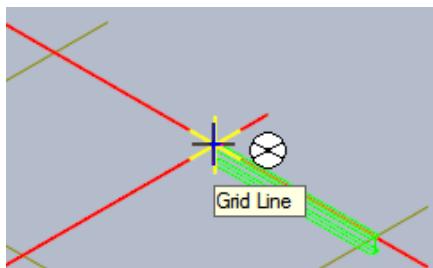
7. Click **End** 🖹 to activate the contiguous placement mode.

8. Specify the end location of the next member.
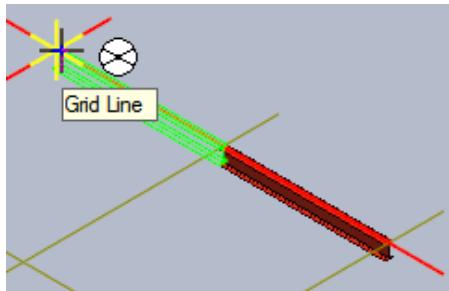
    NOTE To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** ⊗ option is selected.

Place a member using finish mode

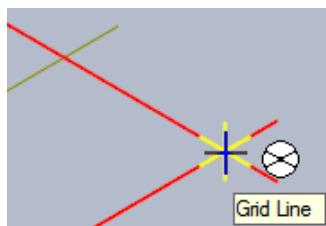1. Click **Place Linear Member System** 📓 on the vertical toolbar.

2. In the **Connection** box, select a frame connection type.

   TIP If you are unsure of which frame connection type to use, review Frame Connections. You can also select the **By Rule** option to allow the software to select automatically a frame connection type.

3. Click **Finish Mode** 🔍 to activate the **Finish** button.

4. Click **Connection Properties** 📑 to activate the **Connection Properties** dialog box.

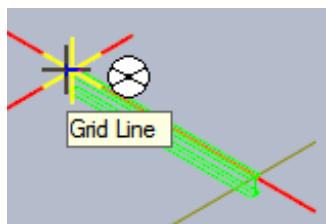5. In the **Type category** box, select the member type category to place.

6. In the **Type** box, select the member type to place.

7. Specify the start location, or first point, of the member.

   TIP You can use the frame connection of another member as the start or end location of the member that you are placing.

8. Specify the end location, or second point, of the member.

9. Edit the start and end frame connection properties in the **Connection Properties** dialog box.

10. Edit the member properties using the **Member Properties** 📑 dialog box.
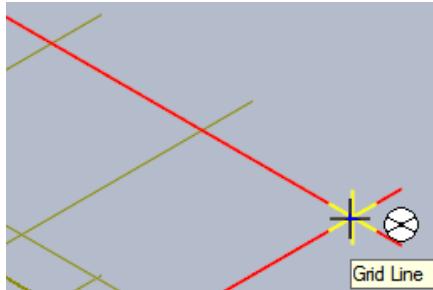
11. Click **Finish**.

NOTES

- To find the intersection of grid lines easily, click **Tools > Options** to verify that the SmartSketch **Intersection** ⊗ option is selected.

- You can set the beta angle relative to your rotated coordinate system by activating **PinPoint** 🔹 and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Edit member system properties

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Members System** in the **Locate Filter**.

3. Select the member system to edit.

4. Click **Edit > Properties**.

5. Edit the properties as needed. For more information about properties, see *Member System Prismatic Properties Dialog Box* (on page 104).

Edit member part properties

1. Click **Select** on the vertical toolbar.

2. Select **Member Parts** in the **Locate Filter**.

3. Select the member part to edit.

4. Click **Edit > Properties**.

5. Edit the properties as needed. For more information about properties, see *Member Part Prismatic Properties Dialog Box* (on page 109).

Edit a frame connection

1. Click **Select** on the vertical toolbar.

2. Select **Frame Connections** in the **Locate Filter**.

3. Select the frame connection to edit.

4. Edit the frame connection as needed. For more information about frame connection properties, see *Frame Connection Properties Dialog Box* (on page 142).

Delete a member system

1. Click **Select** on the vertical toolbar.

2. Select **Members System** in the **Locate Filter**.

3. Select the members to delete.

4. Click **Delete** .

NOTES

- All loads and boundary conditions placed in the Structural Analysis task on the deleted member system are also deleted. This could affect any Analytical Models that have been exported.

- All footings associated with the member system are also deleted.

- Ladders or stairs using the deleted member system as the defined top edge are sent to the **To Do List**.

Convert a member part

1. Click **Select** on the vertical toolbar.

2. Select **Member Parts** in the **Locate Filter**.

3. Select a member part to have its own member system.

4. Click **Convert** on the ribbon.

Modify the cardinal point of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select the member part to modify.
4. Using the **Cardinal Point** control on the ribbon bar, select a new cardinal point for the member part.

NOTE There are 15 cardinal positions available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13.

Modify the end releases of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Member Properties** on the ribbon.
5. Select the **Member Part** tab.
6. In the **Category** box, select **End Releases**.
7. Modify the member part end releases as required.

Modify the angle of a member

1. Click **Select** on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Angle** control on the ribbon bar, edit the member angle.

    TIP You can set the beta angle relative to your rotated coordinate system by activating

    PinPoint and selecting the rotated coordinated system as the active coordinate system in the **Coordinate System** box.

Modify the cross-section of a member

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Section name** box on the ribbon bar, type the new cross-section name for the member, or select **More** to select the cross-section from the Catalog.

Modify the material of a member

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Material** property.

Modify the material grade of a member

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Click **Edit > Properties**.
5. Select the **Cross Section** tab.
6. Select a new value for the **Grade** property.

Modify the type of member

1. Click **Select** ⬚ on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Select a member to modify.
4. Using the **Type Category** and **Type** controls on the ribbon bar, change the type of the member part.

NOTE If the member type category or type that you want to use is not available, you can add it by editing the **Structural Member Type** sheet in the **AllCodeLists.xls** workbook and bulkloading it into the Catalog database. For more information on editing the workbook and bulk loading, see the *Smart 3D Reference Data Guide*.

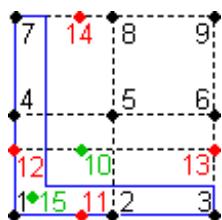# Move a member

1. Click **Select** ⌖ on the vertical toolbar.

2. Select **Member Parts** in the **Locate Filter**.

3. Select a member to move.

4. Click **Move** ✢ on the main ribbon.

5. Define the first point of a vector used to move the member.

6. Define the second point of the vector.

NOTE The frame connections might prevent you from moving the member as you would like. For example, a supported member has a seated frame connection to a supporting member. The seated frame connection prevents you from moving the supported member off of the supporting member. You can, however, slide the supported member along the supporting member as long as the seated frame connection is still valid. Consider copying and pasting a member that you want to move, and then deleting the original.

Move one end of a member

1. Click **Select** ⌖ on the vertical toolbar.

2. Select **Frame Connections** in the **Locate Filter**.

3. Select the frame connection that is near the member end to move.

4. In the **Connection** option, select **By Rule**.

5. Define the new location for the member end.
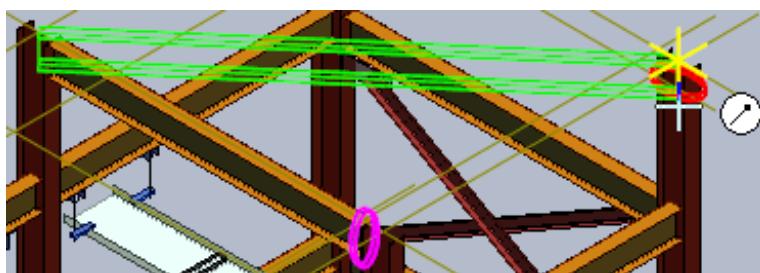
Member System Prismatic Properties Dialog Box

Specifies the properties for the standard member or designed member that you are editing. For an explanation of the difference between a standard member and a designed member, see Members.

Member System Tab (Member System Prismatic Properties Dialog Box) (on page 104)
Relationship Tab (on page 106)
Configuration Tab (on page 37)
Notes Tab (on page 39)

Member System Tab (Member System Prismatic Properties Dialog Box)

Specifies the properties for the member system.

Category

Select the properties to view for the member system.

Standard

Name

Displays the name of the member system. The member system name is based on the **Name Rule** selection. If you want to type a new name for the member system, in the **Name Rule** box, select **User Defined**, and then type a name for the member system in the **Name** box. If you change the **Type Category** of the member system after defining a custom name, the name resets to the default name for the new member type.

Name Rule

Specify the naming rule to use to name this member system.

MemberSystemTypeNameRule - Names the member system using this method: <member type>-<location>-<index number> where <member type> is the selected **Type** property, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Purlin-1-0003.

Default Name Rule - Names the member system using this method: MemberSystem-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberSystem-1-0045.

Unique Name Rule - Names the member system using this method: <parent system>-MemberSystem-<location>-<index number> where <parent system> is the name of the parent system that the Member System belongs to, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Structure System-MemberSystem-1-0001.

User Defined - Select this name rule to name the member system yourself using the **Name** box.

Parent System

Select the system to which the member system that you are placing belongs.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Continuity Type

Indicates the way in which the object system reacts when the system intersects another object system (your automatic splitting preference). The list is defined by the StructContinuity codelist.

Continuous indicates that the object system should split the other object system.

Intercostal indicates that the object system should be split by the other object system.

You cannot split objects that have a **Continuity Type** setting of **Continuous**.

Continuity Priority Number

Specifies the priority when two profile systems that have the same **Continuity** setting intersect. The profile system with the lowest continuity priority number penetrates the other system. For example, two profile systems intersect and the **Continuity** setting of each set to **Continuous**. The profile system with the lowest **Continuity Priority** number splits the other profile system. You can only modify this property at the root system.

Align

Indicates whether the software copies offsets from the frame connection at the member system end to the unsupported frame connection at the other member system end.

True indicates that Smart 3D copies the offsets.

False indicates the Smart 3D does not copy the offsets.

Start East / **Start X**

Displays the X-coordinates of the start of the member relative to the active coordinate system.

Start North / **Start Y**

Displays the Y-coordinates of the start of the member relative to the active coordinate system.

Start Elevation / **Start Z**

Displays the Z-coordinates of the start of the member relative to the active coordinate system.

End East / **End X**

Displays the X-coordinates of the end of the member relative to the active coordinate system.

End North / **End Y**

Displays the Y-coordinates of the end of the member relative to the active coordinate system.

End Elevation / **End Z**

Displays the Z-coordinates of the end of the member relative to the active coordinate system.

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.

- For plate and profile system part properties, parent systems are shown.

- For assembly connection properties, all connected objects are shown.

- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.

- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more

information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

NOTE You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

New location

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

NOTE We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

Notes Tab

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

NOTE Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

Key point

Specifies the key point on the object to which you want to add a note.

Notes at this location, listed by name

Lists all notes for the selected key point on the object.

Date

Displays the date that the note was created. The system automatically supplies the date.

Time

Displays the time that the note was created. The system automatically supplies the time.

Purpose of note

Specifies the purpose of the note.

Author

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

Note text

Defines the note text. The software does not limit the length of the note text.

Show dimension

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

New Note

Creates a new note on the object.

Standard Note

Displays a list of standard notes from which you can select. This feature is not available in this version.

Highlight Note

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

Delete Note

Deletes the currently displayed note.

Member Part Prismatic Properties Dialog Box

Specifies the properties for the member part of the standard member that you are editing. For an explanation of the difference between a member system and a member part, see Members.

Member Part Tab (Member Part Prismatic Properties Dialog Box)

Specifies the properties for the member.

Category

Select the properties to view for the member. Member part properties are divided into several different categories: **Standard**, **Weight and CG**, **Fabrication and Construction**, **Surface Treatment and Coating**, **Responsibility**, and **End Releases**. You select which category that you want to define values for by using the **Category** option.

Standard

Name

Displays the name of the member part. The member part name is based on the **Name Rule** selection. If you want to type a new name for the member part, in the **Name Rule** box, select **User Defined**, and then type a name for the member part in the **Name** box.

Name Rule

Specify the naming rule to use to name this member part.

- **Member Part Type Name Rule** - Names the member part using this method: <member type>-<location>-<index number> where <member type> is the selected **Type** property, <location> is the global workshare location ID, and <index number> is a unique index number that starts at 0001. For example: Beam-1-0003.

- **Default Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.

- **Unique Name Rule** - Names the member part using this method: MemberPartPrismatic-<location>-<index number> where <location> is the global workshare location ID and <index number> is a unique index number that starts at 0001. For example: MemberPartPrismatic-1-0045.

- **User Defined** - Select this name rule to name the member system yourself using the **Name** box.

NOTE If you plan to export this model to Tekla Structures using CIMsteel, you must know that Tekla limits part names to 21 characters. If you use a Smart 3D naming rule that creates a member part name that is longer than 21 characters, Tekla truncates the name during the CIMSteel import into Tekla. This truncation can cause all your member parts to have the same name in Tekla because Smart 3D naming rules have the unique index at the end of the name.

Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the

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Catalog task.

Type

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Length

Displays the length of the member without cutbacks applied. You cannot change this value.

Cut Length

Specifies the length of the member with cutbacks applied. You cannot change this value. A cutback is that part of a member removed by an assembly connection or by a manually placed trim (a cope, for example).

Reporting Requirements

Specify whether this object is reported.

Reporting Type

Select the reporting requirements code for the object. Valid codes are defined in Catalog in the **Reporting Type** select list.

Piece Mark

Specifies the piece mark of the member. Piece marks are typically used when importing and exporting member through CIMsteel. If a piece mark displays, the mark is usually the identity of the manufactured part from the other software package.

Assembly Mark

Specifies the assembly mark of the member. If the member was imported through CIMsteel, the assembly mark is usually the identity of the manufactured assembly to which this member belongs.

Seam Angle

Defines the location of the seam. This angle is measured clockwise from orientation vector of the member looking down the member from the start end.
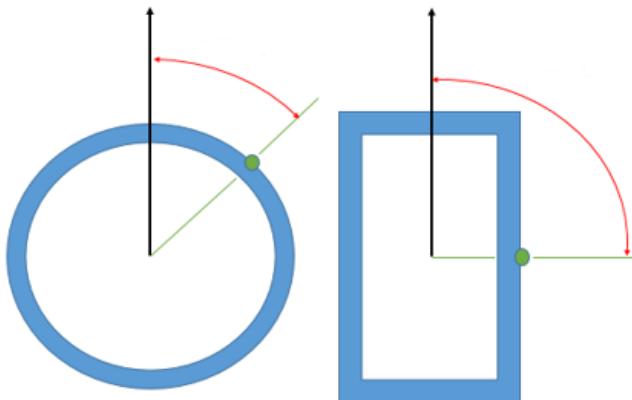
Seam Angle is only available on cross-sections that use the **Has Seam** definition. Refer to the *Smart 3D Symbol 2D User's Guide* and *Symbol 2D Reference Data Guide* for more information about cross-sections.

Weight and CG

Displays the center-of-gravity and the weight of the selected object. The center-of-gravity locations are displayed relative to the active coordinate system along the X-, Y-, and Z-axes. The weight value that is displayed in the properties dialog box is calculated as the material density multiplied by the object's solid volume. Therefore, the material of the object affects the weight value that is displayed here. Check the material assigned to the object if the weight displayed is an improbable value. For the most accurate weight calculation, use the **Tools > Run Reports** command.

Dry Weight

Specifies the dry weight of the object.

Wet Weight

Specifies the wet weight of the object.

NOTE For equipment, the **Weight and CG** property **Wet Weight** is the sum of **Dry Weight** and **Water Weight**. The dry weight and water weight values are catalog properties entered on the part sheet for the equipment.

Dry CG X

Specifies the X-axis location of the dry center-of-gravity.

Dry CG Y

Specifies the Y-axis location of the dry center-of-gravity.

Dry CG Z

Specifies the Z-axis location of the dry center-of-gravity.

Wet CG X

Specifies the X-axis location of the wet center-of-gravity.

Wet CG Y

Specifies the Y-axis location of the wet center-of-gravity.

Wet CG Z

Specifies the Z-axis location of the wet center-of-gravity.

Dry WCG Origin

Specifies the way in which the dry weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the dry weight center-of-gravity location relative to the active coordinate system.

Wet WCG Origin

Specifies the way in which the wet weight center-of-gravity location is defined. The list is defined by the WCGOrigin codelist.

Computed indicates that the software calculates the origin location.

Defined indicates that you want to manually define the wet weight center-of-gravity location relative to the active coordinate system.

Fabrication and Construction

Fabrication Requirement

Specifies the fabrication requirement for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Fabrication Type

Specifies the type of fabrication for the object. To change the options on the list, edit the **Fabrication Type** select list in Catalog.

Construction Requirement

Specifies the construction requirement for the object. To change the options on the list, edit the **Construction Requirement** select list in Catalog.

Construction Type

Specifies the type of construction for the object. To change the options on the list, edit the **Construction Type** select list in Catalog.

Surface Treatment and Coating

Exterior Coating Requirement

Specifies the coating requirement for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Exterior Coating Type

Specifies the type of coating for the object. To change the options on the list, edit the **Coating Type** select list in Catalog.

Coating Color

Specifies the color of the object coating. To change the options on the list, edit the **Coating Color** select list in Catalog. Smart 3D includes this property in the painting area report.

Exterior Coating Area

Specifies the area of the coating for the object.

Responsibility

Cleaning Responsibility

Specifies the party responsible for cleaning the object. To change the options on the list, edit the **Cleaning Responsibility** select list in Catalog.

Design Responsibility

Specifies the party responsible for designing the object. To change the options on the list, edit the **Design Responsibility** select list in Catalog.

Fabrication Responsibility

Specifies the party responsible for fabricating the object. To change the options on the list, edit the **Fabrication Responsibility** select list in Catalog.

Installation Responsibility

Specifies the party responsible for installing the object. To change the options on the list, edit the **Installation Responsibility** select list in Catalog.

Painting Responsibility

Specifies the party responsible for painting the object. To change the options on the list, edit the **Painting Responsibility** select list in Catalog.

Requisition Responsibility

Specifies the party responsible for ordering the object. To change the options on the list, edit the **Requisition Responsibility** select list in Catalog.

Supply Responsibility

Specifies the party responsible for delivering the object. To change the options on the list, edit the **Supply Responsibility** select list in Catalog.

Testing Responsibility

Specifies the party responsible for testing on the object. To change the options on the list, edit the **Testing Responsibility** select list in Catalog.

End Releases

Start Member Release

Select the directions to release at the start of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions is not available, select the **User Defined** option and define the releases yourself.

Start X Displacement

Defines if the X direction at the start of the member part is fixed or free.

Start Y Displacement

Defines if the Y direction at the start of the member part is fixed or free.

Start Z Displacement

Defines if the Z direction at the start of the member part is fixed or free.

Start X Rotation

Defines if the X moment direction at the start of the member part is fixed or free.

Start Y Rotation

Defines if the Y moment direction at the start of the member part is fixed or free.

Start Z Rotation

Defines if the Z moment direction at the start of the member part is fixed or free.

End Member Release

Select the directions to release at the end of the member part. Directions are defined in the local coordinate system of the member system. If the combination of directions is not available, select the **User Defined** option and define the releases yourself.

End X Displacement

Defines if the X direction at the end of the member part is fixed or free.

End Y Displacement

Defines if the Y direction at the end of the member part is fixed or free.

End Z Displacement

Defines if the Z direction at the end of the member part is fixed or free.

End X Rotation

Defines if the X moment direction at the end of the member part is fixed or free.

End Y Rotation

Defines if the Y moment direction at the end of the member part is fixed or free.

End Z Rotation

Defines if the Z moment direction at the end of the member part is fixed or free.

See Also

Member Part Prismatic Properties Dialog Box (on page 109)

Cross Section Tab (Member Part Prismatic Properties Dialog Box)

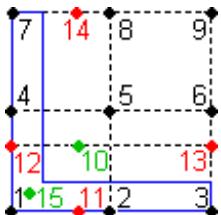Specifies the properties for the cross-section of a standard member part.

Section Standard

Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Member Section Name

Defines the cross-section for the member. If you know the section name, type it in. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type W10X* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. See Structure Reference Data Guide for more information about reference data.
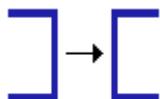
Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option would be when you want the flanges of a channel section to point in the opposite direction. The **Reflect** property is not available when editing members that have **Seated**, **Flush**, or **Centerline** frame connections.

Material

Select a material for the member. Materials are defined in the **AllCommon.xls** workbook or in the Catalog task.

Grade

Select a material grade for the member. Material grades are defined in the **AllCommon.xls** workbook or in the Catalog task.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.
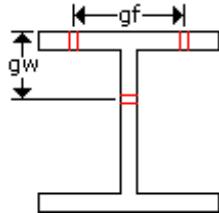
Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Flange Gage (gf)

Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Gage (gw)

Displays the bolt gage for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Section Name

Specifies the section name. This name displays when you label members. This property is read-only. To edit this value, edit the cross-section's properties in Catalog or in the corresponding workbook.

Short Name

Displays the short name for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Alternate EDI Name

Displays the Electronic Data Interchange name for the section. This name is used when translating sections through CIMsteel. This property is currently not used.

Description

Specifies the description. This property is read-only. To edit this value, you need to edit the cross- section's properties in Catalog or in the corresponding workbook.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Theoretical Maximum Yield Stress (Fy''')

Displays the maximum yield stress for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Point in Flange (Qf)

Displays the first moment of area for the flange. This property is read- only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Statical Moment at Mid Depth of Section (Qw)

Displays the first moment of area for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Normalized Warping Function (Wno)

Displays the normalized warping function. The function is defined in square inches or square millimeters. This property is read-only. To edit this value, you need to edit the cross- section's

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properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X1)

Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Beam Buckling Factor (X2)

Displays the beam buckling factor. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Thickness (tw)

Displays the web thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Depth (d)

Displays the web depth. This property is read-only. To edit this value, you need to edit the cross- section's properties in the Catalog task or in the corresponding workbook.

Flange Thickness (tf)

Displays the flange thickness for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Flange Width (bf)

Displays the width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdetail)

Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Distance to Web Toe Fillet (kdesign)

Displays the distance from the outer face of the flange to the web toe of the fillet of the rolled shape or the equivalent distance on the welded section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Group Id

Displays the material group identification for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Designed Member Properties Dialog Box

Specifies the properties for the designed member that you are editing. For an explanation of the difference between a standard member and a designed member, see Members.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

Occurrence Tab (Designed Member Properties Dialog Box)

Specifies the properties for the select designed member.

Category

Select the properties that you want to view for the member. Only **Standard** properties are available for designed members.

Show Dimensional Legend

Displays a graphic indicating the different dimensions that can be specified for the selected designed member.

Standard

Name

Displays the name of the designed member. The designed member name is based on the **Name Rule** selection. To type a new name for the designed member, select **User Defined** in the **Name Rule** box, and then type a name for the designed member in the **Name** box.

Name Rule

Specify the naming rule to use to name this designed member. You can select one of the listed rules or select **User Defined** and type the designed member name in the **Name** box.

Parent System

Specifies the name of the parent system. You can define new systems in the Systems and Specifications task.
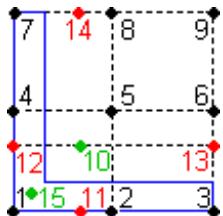
Section Standard

Specifies the section library from which you selected the section name. Sections are defined in the reference data.

Member Section Name

Defines the cross-section for the member. If you know the section name that you want, you can type it. You can use the asterisk [*] character wildcard to see all sections that contain that text. For example, type BUIH* to see all W10X sections in the catalog. Select **More** to browse the catalog for the section to use. Sections are defined in the reference data. For more information about reference data, see *Structure Reference Data Guide.*
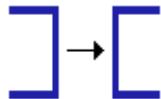
Cardinal Point

Displays the relative position of the structural cross-section to the member placement line. Nine cardinal positions (1 -9) are available. The location of cardinal points 10 (center-of-gravity) and 15 (shear center) depend on the section shape. The local z-axis of the member and the center-of-gravity point of the section define cardinal points 11 and 14. The local y-axis of the member and the center-of-gravity point of the section define cardinal points 12 and 13. Cardinal points 10 through 15 are unavailable for designed members or cans.

Angle

Defines the angle by which the section is rotated about the member axis.

Reflect

Reflects or mirrors the cross-section about the member's local z-axis. This parameter affects both symmetric and asymmetric sections. An example of when to use this option is when you want the flanges of a channel section to point in the opposite direction.

Type category

Specifies the type category of the member, such as a beam or a column. The available member properties change depending on the member type category that you select. You can define a custom member type category by editing the **Structural Member Type** list in the Catalog task.

Type

Specifies the type of member, such as a beam or column. The available member properties change depending on the member type that you select. This property is a hierarchical child of **Type Category**. To change the options on the list, edit the **Structural Member Type** select list in Catalog.

Priority

Specifies the priority assigned to the object system. Smart 3D uses the priority to group objects. To change the options on the list, edit the **Structural Member Priority** select list in Catalog.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric about X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.

Is Symmetric about Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Cross Section Tab (Designed Member Properties Dialog Box)

Specifies the properties for the cross-section of a designed member.

Name

Displays the cross-section name for the designed member. Sections are defined in the reference data. For more information about reference data, see *Structure Reference Data Guide*.

Centroid X

Displays the location of the centroid along the local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Centroid Y

Displays the location of the centroid along the local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about X (Ixx)

Displays the moment of inertia for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Moment of Inertia about Y (Iyy)

Displays the moment of inertia for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Statical Moment (Sw)

Displays the warping statical moment. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about X (Sxx)

Displays the section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Elastic Section Modulus about Y (Syy)

Displays the section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Torsional Moment of Inertia (J)

Displays the torsional moment of inertia for the section. This property is read- only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Warping Constant (Cw)

Displays the warping constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Flexural Constant (H)

Displays the flexural constant for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Polar Radius of Gyration about Shear Center (ro)

Displays the polar radius of gyration about the shear center. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about X axis (Rxx)

Displays the radius of gyration for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Y axis (Ryy)

Displays the radius of gyration for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Radius of Gyration about Principle XY (Rxy)

Displays the radius of gyration about the principle xy-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about X (Zxx)

Displays the plastic section modulus for the section's local x-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Plastic Section Modulus about Y (Zyy)

Displays the plastic section modulus for the section's local y-axis. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Is Hollow

Displays **True** if the cross-section is hollow, such as a tube or can. Displays **False** if the cross-section is solid, such as a bar, or open, such as an I-beam.

Is Symmetric About X

Displays **True** if the cross-section is symmetric about the local x-axis of the cross-section.
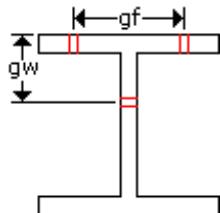
Is Symmetric About Y

Displays **True** if the cross-section is symmetric about the local y-axis of the cross-section.

Flange Gage (gf)

Displays the bolt gage for the flange. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Web Gage (gw)

Displays the bolt gage for the web. This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Plastic Neutral Axis (xp)

Displays the distance from the designated edge of the cross-section's plastic neutral axis (PNA) along the local x-axis.

Plastic Neutral Axis (yp)

Displays the distance from the designated edge of the cross-section's plastic neutral axis (PNA) along the local y-axis.

Section Name

Specifies the section name. This name displays when you label members. This property is read-only. To edit this value, edit the cross-section's properties in Catalog or in the corresponding workbook.

Area

Specifies the cross-section area for the object. This property is read-only. To change this value, edit the object properties in Catalog or in the corresponding workbook.

Depth

Displays the depth for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Width

Displays the flange width for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Perimeter

Displays the outside perimeter distance for the section. This property is read-only. To edit this value, you need to edit the cross-section's properties in Catalog or in the corresponding workbook.

Unit Weight

Displays the weight of the section. The unit weight is defined in mass per length pound per foot (lbm/ft). This property is read-only. To edit this value, you need to edit the cross-section's properties in the Catalog task or in the corresponding workbook.

Nonlinear Plate System Properties Dialog Box

Specifies the properties for the root or leaf plate system of the designed member that you are editing. A designed member contains multiple plate systems that you can individually edit.

OK

Accepts the values, and closes the dialog box.

Cancel

Rejects the values, and closes the dialog box.

Apply

Accepts the values, but does not close the dialog box.

NOTES

- A leaf system always shares the same property values as its root system upon creation.

- When a root and leaf system have properties with the same values, a change to a root system value changes the leaf system value.

- When you change the value of a property of a leaf system, later changes to that property on the root system do not change the leaf system value.

- When a root system is split, new leaf systems are created that inherit the property values of the root system, *except* for values on the original leaf system that are changed before the split.

- Some leaf system properties cannot be changed at the leaf system level. They must be changed at the root system level.

Main Tab (Nonlinear Plate System Properties Dialog Box) (on page 128)
Material Tab (Nonlinear Plate System Properties Dialog Box) (on page 131)
Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member) (on page 132)
Relationship Tab (on page 106)
Configuration Tab (on page 37)
General Tab (on page 136)

Main Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the general properties of the nonlinear extruded plate system.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide.*

Rule

Select the naming rule to use to name the plate system.

StdPlateSystemNamingRule - Uses syntax based on the reference plane.

- Plate system on a reference plane: *<Reference plane name>-<Index number><Plate type>-<Workshare location ID>*. For example, F383-1TBH-1, where F383 is the reference plane name, 1TBH is an index number appended by the Plate Type (Transversal Bulkhead in this example), and the final 1 is the Workshare Location ID. Child plates

inherit the parent name and add a unique index number to the end, such as F383-1TBH-1-103.

- Plate system not created on a reference plane, or on a standalone plane (created from a copy or model data reuse operation): *<Global CS axis><Global CS position along the axis in mm>-<Index number><Plate type>-<Workshare location ID>*, as shown in the following examples.

  Global CS Location: 2 m
  XY Plane at Global Z Location: Z2000-1DCK-1
  ZX Plane at Global Y Location: Y2000-1LBH-1
  YZ Plane at Global X Location: Y2000-1TBH-1

  Global CS Location: 2.75 m
  XY Plane at Global Z Location: Z2750-1DCK-1
  ZX Plane at Global Y Location: Y2750-1LBH-1
  YZ Plane at Global X Location: Y2750-1TBH-1

    - Plate system not orthogonal to any major plane: *A-<Unique index number><Plate type>-<Workshare location ID>*, such as A-202DCK-1.

    - Standalone planes used as boundaries, created from a copy or model data reuse operation: *Plane: <Global CS axis> = <Global CS position along the axis>*, such as Plane: Z = 2500mm. The name only displays in the **Boundary List** Dialog Box.

  StdHierarchyChildNamingRule - Uses the following syntax: *<Parent system name>-<Object type>*. For example, Model_MDB-IJPlate1, where Model_MDB is the parent system, IJPlate is the object type, and 1 is an index number appended to the object type.

  User Defined - Allows you to type any name.

Type

Specifies the type of plate system that you are placing. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

If you do not specify a type, the software automatically determines the type based on the plate system orientation. Plate systems that are mostly horizontal (XY plane) are assigned to **Deck**. Plate systems that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate systems that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. If you are using material handling mode, plate systems are assigned to **General Plate**.

NOTE The naming rule also uses **Type** to name the plate system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Specifies a parent system for the plate system. You can define parent systems in the Systems and Specifications task. When you create a plate system, the software uses the property values of the parent system as the initial property values for the plate system. When a parent

property value changes, the corresponding child property value also updates.

If this plate system is a bracket system, you cannot select another root plate system as the parent system because a bracket system is a root plate system.

Surface Geometry Type

Displays the Molded Forms command used to create the plate system.

Specification

Defines the structural specification for the plate system. This property can only be modified at the root system.

Description

Defines an optional description for the plate system.

Continuity

Specifies the continuity type for the plate system. Continuity defines how the plate system reacts when it intersects another plate or profile system. Select **Continuous** to indicate that the plate system penetrates the other system. Select **Intercostal** to indicate that the plate system is penetrated by the other system. This property can only be modified at the root system.

Split Priority

Specifies the continuity priority. This priority is used to specify which plate system is continuous and which penetrated (split) when two plate systems intersect, but have the same value for **Continuity**. Plate systems with a lower continuity priority (such as 1, 2, or 3) penetrate plate systems with a higher continuity priority (such as 7, 8, or 9). This property can only be modified at the root system.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Tightness

Specifies the level of tightness as it applies to the entire plate system.

Active

Determines whether a leaf system is active and applicable for modeling, drawing, and reporting operations. To exclude the leaf system from these operations, select **No.** In the **Workspace Explorer**, the lock icon 🔒 is shown over the icon of the deactivated leaf system, and the detailed or light part associated with the leaf system is also deleted. In addition, because connections to deactivated leaf system are not valid, you must resolve such invalid connections in the **To Do List**. To change the leaf profile system back to the active state, select **Yes**; however, to have the detailed part, you must detail the leaf system explicitly. For information on leaf systems, see *Marine Structure Hierarchy in the Workspace Explorer* in the *Molded Forms User's Guide* or in the *Common User's Guide.*

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:
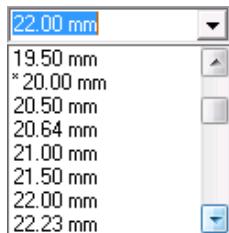
This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Material Tab (Nonlinear Plate System Properties Dialog Box)

Specifies the material properties for the nonlinear extruded plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:
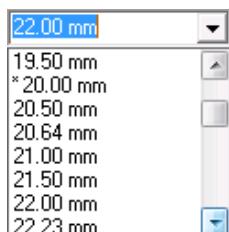
This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Molded Conventions Tab (Nonlinear Plate System Properties Dialog Box - Designed Member)

Specifies the plate orientation rules for the plate system of the designed member that you are editing.

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. A plate on a designed member can be **Flange Plate**, **Web Plate**, or **Tube Plate**. Plate system types are defined by the designed member cross-section. You cannot change the type.

Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.

Out - Positions thickness away from the center of the member.

Up - Positions thickness in the positive v-axis direction.

Down - Positions thickness in the negative v-axis direction.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.

Out - Orients profiles away from the center of the member.

Down - Orients profiles in the negative v-axis direction.

Up - Orients profiles in the positive v-axis direction.

Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.

Out - Orients profile flanges away from the member plate web.

Left - Orients profile flanges in the negative u-axis direction.

Right - Orients profile flanges in the positive u-axis direction.

Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.

Towards End - Orients profile flanges in the direction of the end point of the designed member.

Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.

Out - Positions thickness away from the center of the member.

Left - Positions thickness in the negative u-axis direction.

Right - Positions thickness in the positive u-axis direction.

Centered - Centers thickness about the molded surface. This is the default value for designed members.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.

In - Orients profiles towards t center of the member.

Left - Orients profiles in the negative u-axis direction.

Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.

Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.

Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.

Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.

Towards End - Orients profile flanges in the direction of the end point of the designed member.

Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.

Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite

direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.

Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.

Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

▪ Are on the plate of a designed tube member

▪ Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.

Towards End - Orients profile flanges in the direction of the end point of the designed tube member.

Axial Profiles

Defines the orientation of the flanges for profiles that:

▪ Are on the plate of a designed tube member

▪ Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.

Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.

General Tab

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You can edit the properties on these tabs.

Plate Part Properties Dialog Box

Specifies the properties for the plate part that you are editing.

Main Tab (Plate Part Properties Dialog Box)

Specifies the general properties of the plate part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide.*

Rule

Select the naming rule to use to name the object.

PlatePartRule - Creates a name based on the parent root plate system name, the location within the parent root plate system, and the parent assembly name.

User Defined - Name the plate part yourself by using the appropriate box.

Type

Displays the type of the plate part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Displays the parent system for the object.

Surface Geometry Type

Displays the Molded Forms command used to create the root parent plate system. It is inherited from the parent system.

Specification

Specifies the specification for the object. This property is inherited from the parent system.

Description

Specifies a description for the object.

Tightness

Specifies the water tightness of the object. It is inherited from the parent system.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Material Tab (Plate Part Properties Dialog Box)

Displays the material properties for the plate part that you are editing. The material properties are inherited from the parent system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Actual Thickness

Displays the material thickness for the object.

General Tab (Plate Part Properties Dialog Box)

Specifies general properties of the plate part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Plate Part Properties Dialog Box)* (on page 137).

PlateTightness

Specifies the water tightness of the plate part. The list is defined by the StructPlateTightness codelist.

PlateType

Specifies the type of the plate part. The list is defined by the StructPlateType codelist.

Thickening Technique Requested

Specifies the technique requested to thicken the plate part.

This property affects only the behavior of plate parts derived from systems. The property defaults to **Procedural** at creation. Changing the value of this property can affect the geometry of the part as described below.

Standalone plate parts initially display nothing for this property. You can change the value, but those changes do not change the part or its geometry. Smart 3D thickens all standalone plate parts, including collars, using procedural thickening.

Undefined (blank) indicates that Smart 3D selects the appropriate technique. Smart 3D automatically changes this value to **Procedural**.

Procedural indicates the part is thickened using the standard thickening method provided by the ACIS modeler. This works for the vast majority of parts, and you should not change this value unless there is a specific reason to try a different technique.

NOTE If the thickening fails using ACIS, and the part is a hull part, the software automatically tries the **NonProcedural** thickening technique. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Procedural with slit is used for parts that have internal slits or discontinuities in the surface of the system that form part of the boundary of parts on the system. The currently-supported example of this is a surface with a combination of split and bent knuckles joined end to end. Because Smart 3D does not support splitting knuckles and changing the properties of different segments, these surface must be constructed with the correct geometry to reflect the bent and split portions. Parts that are bounded by the slit, and potentially parts on the system that are within half a meter of the slit, must have this value set to assure a proper trim. Setting this attribute on parts that do not need it should not cause a failure but may cause the part to take more time to detail. The following pictures show an example of this type of surface, first the full surface and then a close-up in the region of the slit or discontinuity.

The plate system in these pictures is split into three parts by the knuckle (shown as white) and the two black seams. You must be sure that a single part does not touch both sides of the slit.
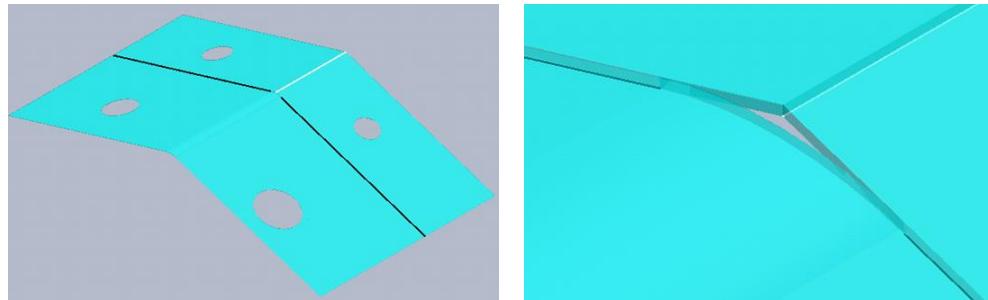
NonProcedural is an infrequently used option. Historically, a much higher percentage of parts failed to thicken with the ACIS modeler, and many designers used a specific hull form definition tool that produced a surface with very specific characteristics. **NonProcedural** thickening provided a tool that would thicken almost any part on this type of surface, even if the ACIS modeler failed. The ACIS modeler has improved greatly, and the **NonProcedural** thickening method has not been enhanced to handle the surfaces produced by many of the hull form tools currently in use. Setting this option may solve a failure-to-thicken problem on a hull part.

NOTE If **NonProcedural** thickening fails for a hull shell part, Smart 3D automatically tries to thicken the part using **Procedural** thickening. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Thickening Technique Used

Displays the technique used to thicken the plate part. This is blank for a standalone plate part, although **Procedural** is always used for these parts. It displays one of the values defined for **Thickening Technique Requested** for a system-derived part. The meaning for these values is the same as defined for **Thickening Technique Requested**.

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Weight & CG Tab

Displays the center-of-gravity and the weight of the part.

Type

Displays the type of weight: **Dry Weight** or **Wet Weight**.

Weight

Displays the weight of the plate part pertaining to the weight type.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**.

CoG Coordinate System

Specifies the coordinate system to use to measure the center of gravity.

Extended User Attributes Tab

Displays the user attributes that have been bulkloaded on the report data object related to the part. Changes to these attributes do not cause the related manufacturing part to go out-of-date. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Routing Tab

Specifies routing properties for the part. By default, no routing properties are set.

Create

Sets routing properties for the part.

Delete

Deletes the routing properties for the part.

Rule

Specifies a predefined rule set or a user-defined rule that is applicable to the part.

Workcenter

Select an assembly workcenter where the part is produced from a facility-defined rule set. The list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Stage Code

Specifies the workcenter code.

No. Of Actions

Specifies the number of actions required to produce the part.

Action

Specifies the name of the action.

Machine

Specifies the type of machine used to perform the required operation.

Code

Specifies the machine code.

Frame Connection Properties Dialog Box

Specifies the properties for the frame connection that you are editing.

See Also

General Tab (Frame Connection Properties Dialog Box)

Specifies the properties for the frame connection.

TIP The *supported* member is the member that you are placing. The *supporting* member is the existing member in the model to which you are connecting.

Category

Select the type of properties to view for the selected frame connection.

Type

Displays the type of frame connection. This box is read-only. See Frame Connections for more information about the different types that are available.

Property

Displays all properties associated with the selected frame connection. The list of available properties depends on what was defined in the reference data for the frame connection type.

Value

Specifies the values for the frame connection properties.

Name

Specifies the frame connection name.

Name rule

Specifies the naming rule used to name the frame connection.

- **Default Name Rule** - Names the frame connection using the format "<frame connection type>-<location>-<index>" where <frame connection type> is what you select in the **Type** box on the ribbon, <location> is the global workshare location ID, and <index> is a unique index number that starts at 0001. For example, Axis-Along-1-0043.

- **User Defined** - Select this rule to name the frame connection yourself using the **Name** box.

Position rule

Defines how the frame connection is to behave when the supporting member is moved. You can select one of three options:

- **Intersection** - The member system lengthens or shortens to maintain the connections with the supporting member. The end of the supported member system slides to a new location on the supporting member.

- **Ratio** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same relative position (that you can define) along the supporting member system. This option is similar to the **Distance** option except that you define a percentage ratio from the supporting member end.

- **Distance** - The member system lengthens or shortens to maintain the connection with the supporting member. The end of the supported member system stays in the same position (that you can define) along the supporting member system. This option is similar to the **Ratio** option except that you define an actual distance from the supporting member end.

Original Position:
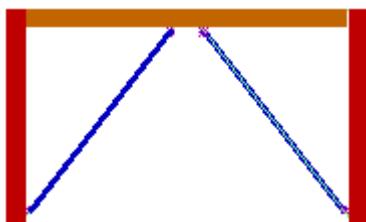
Intersection Position Rule:
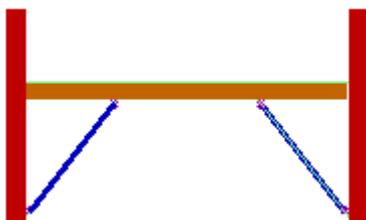
Ratio and Distance Position Rule:
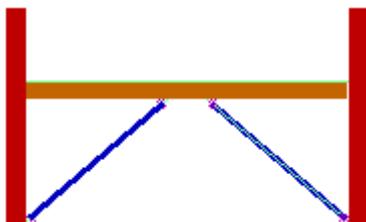
Distance along

Type the distance from the supporting member end that the supported member is positioned. You must include the units of measurement when defining this distance. The supporting member end that is measured from, is defined using the **End** option. This option is available when you set **Position Rule** to **Distance**.

Ratio

Type the ratio of the supporting member length that the supported member is positioned. For example, type **.25** if you want the supported member a fourth of the way along the supporting member. Type **.333** if you want the supported member a third of the way, and so forth. Which supporting member end that is measured from is defined using the **End** option. This option is available when you set **Position Rule** to **Ratio**.

End

Specifies which end of the supporting member that the ratio or the distance-along distance is measured. You can select **Start**, **End**, or **Auto**. Start is the first member end that was placed. End is the second member end that was placed. If you select **Auto**, the software automatically selects the supporting member end that is closest to the frame connection. The **Auto** setting is recommended so that you do not have to worry whether the supporting member was modeled left-to-right, right-to-left, top-to-bottom, or bottom-to-top.

Align Frame Connection Properties

Side Justification

Select the side of the plane on which to place the member.

- **None** - Aligns the member axis so that it is placed on the surface.

- **Seated** - Sets the member on top of the surface. The selected **Orientation Side** is nearest the surface.

- **Center** - The center of the cross-section is placed on the surface. The center is calculated by the top-bottom, left-right bounds of the cross-section shape.

- **Flush** - Uses the alignment surface's top and bottom extent to position the supported member. The supported member typically lies within the body of the alignment plane with one edge of the member flush with the alignment surface but can be offset.

Orientation Control

If **On**, the member angle is controlled by the **Orientation Angle** value defined for the Align

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frame connection. If **Off**, you can set the angle of the member independently of the align surface by using the **Rotation** property for the member part.

Orientation Side

Specifies the side of the supported member's cross-section that is mated to the alignment surface. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

Orientation Angle

When **Orientation Control** is **On**, this angle is added to the selected **Orientation Side** to rotate the member. If the Align frame connection is used on both ends of the member and you define a different angle value for each end, the angle defined at the start end of the member is used.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. An example of when to use this option is when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.

Extend Distance

Specifies the offset distance from the bounding surface along the member's axis.

Side Clearance

Specify the offset distance perpendicular to the align surface to place the member axis.

Lateral Distance

Specifies the offset distance in the plane of the align surface to place the member axis.

Axis Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction after the two cardinal points are aligned.

Y Offset

Specifies an offset to apply in the y-direction after the two cardinal points are aligned.

Z Offset

Specifies an offset to apply in the z-direction after the two cardinal points are aligned.

Coordinate System

Specifies the coordinate system to use for the offset values. Local is the local coordinate system of the supporting member.

Supporting CP

Specifies to which cardinal point on the supporting member system to align the supported member system's cardinal point. You can specify any cardinal point number, or select 0 to use the cardinal point with which the supporting member was placed.

Gap Frame Connection Properties

There are three members in a gap frame connection:

- The *target* member is the member always to move and is the owner of the frame connection. The target member is shown as blue in the figure below.

- The *primary* member is the member to which the other two members are attached. The primary member is shown as red in the figure below.

- The *secondary* member is the third member in the joint. The software does not require the secondary member to be in the same plane as the target member. The secondary member is shown as orange in the figure below.

Actual Gap Distance (G)

Displays the actual gap distance between the target and secondary member. This distance is shown as G in the figures below.

Actual Centerline Distance (X)

Displays the actual gap between the center line of the target member and the center line of the secondary member. This distance is shown as X in the figures below.

Actual Overlap Distance (L)

Displays the distance between the lowest point of the target and secondary member saddle on the primary member to the point where the target and secondary member intersect. This distance is shown as L in the figures below.

Actual Overlap Distance (A)

Displays the amount of overlap between the target and secondary member. This distance is shown as A in the figures below.

Allowable Centerline Distance

Displays the calculation of the primary member diameter times the **Centerline Design Factor**. The centerline distance should be less than or equal to this value.

Allowable Overlap Distance (Fmin $\times$ B)

Specify the minimum overlap distance (A) that is acceptable.

Allowable Overlap Distance (Fmax $\times$ B)

Specify the maximum overlap distance (A) that is acceptable.

Design Passed (Centerline)

Display **True** if the centerline design passed. Displays **False** if the centerline failed. The centerline design pass/fail is calculated by comparing actual centerline distance with the allowable centerline distance.

Design Passed (Overlap)

Display **True** if the overlap design passed. Displays **False** if the overlap failed. The overlap

design pass/fail is calculated by (Min. Factor $\times$ B) < A < (Max. Factor $\times$ B) and then comparing that value to the **Allowable Overlap Distance (Fmin $\times$ B)** and the **Allowable Overlap Distance (Fmax $\times$ B)** values. You define the minimum and maximum design factors using the **Overlap Design Factor (Fmin)** and **Overlap Design Factor (Fmax)** properties. The B distance is shown in the figures below.

Direction

Select the direction to calculate the gap.

- **Axially Along Surface** - Select this mode when you want to define the offset as the distance between the center lines of the target and secondary members. This distance is measured parallel to the primary member centerline.

- **Axially and Flush Along Surface** - **This option is currently not available.** Select this mode when you want to define the offset as the distance between the center lines of the target and secondary members and you want the target member to be flush with the outside of the primary member. The axially distance is measured parallel to the primary member centerline. Use the **Flush Direction** and **Flush Offset** options to refine exact flush placement.

- **Radially Along Surface** - Select this mode when you want to define the offset as the distance between the closest points on the hull of the target and secondary members. This distance is measured around the hull of the primary member.

Offset Required

Type the gap distance that you want between the target and secondary member. Select how you want to define this distance using the **Offset Type** box.

Centerline Distance Roundoff

Type the distance to which the centerline gap should be rounded off.

Centerline Design Factor

Type the ratio of the primary member section size to use in calculating whether the centerline design passed or failed. The default value is **0.25**, which specifies that the primary member diameter be divided by 4 (D/4).

Flush Direction

This option is currently not available. Specify which direction you want the target member to move to become flush with the primary member.

Flush Offset

This option is currently not available. Specify the distance from the primary member hull edge to the target hull edge. An offset of zero indicates that the two member edges are perfectly flush.

Offset Along

Defines if the target member moves along the primary member or along the secondary member to create the gap. This property is only available for Gap Single Axial gap connections.

Offset Type

Select how you want to specify the **Offset Required** value.

- **Gap** - The **Offset Required** distance is between the two closest points of the target member hull and the secondary member hull. This is shown as G in the figures below.

- **Centerline** - The **Offset Required** distance is between the center line of the target member and the center line of the secondary member. This is shown as X in the figures below.

- **Overlap** - The **Offset Required** distance is between the lowest point of the target and secondary member saddle on the support member to the point where the target and secondary member intersect. This is shown as L in the figures below.

Overlap Design Factor (Fmin)

Defines the minimum design factor for the calculation: Min $\times$ B < A < Max $\times$ B. The default value is 0.3. This calculation is used to determine if **Design Passed (Overlap)** passes or fails. The A and B distances are shown in the figures below.

Overlap Design Factor (Fmax)

Defines the maximum design factor for the calculation: Min $\times$ B < A < Max $\times$ B. The default value is 0.5. This calculation is used to determine if **Design Passed (Overlap)** passes or fails. The A and B distances are shown in the figures below.

Axially Along Surface:
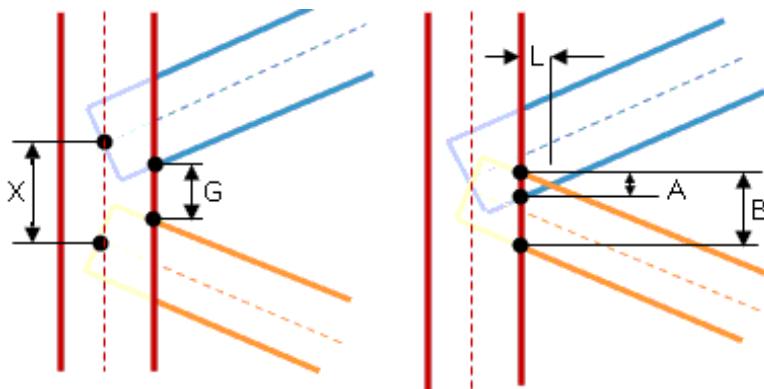
Radially Along Surface:

Seated, Flush, and Centerline Frame Connection Properties

Side

Select the side of the supporting member on which you want to place the supported member.

Offset

Specify the distance to place the supported member from the supporting member. For seated and flush frame connections, the offset is between the side of the supporting member that you specified with the **Side** option and the supported member's side that you specify with the **Edge** option. For centerline frame connections, the offset is between centerline of the supporting member and the supported member's side that you specify with the **Edge** option.

Edge

Specifies the side of the supported member's cross-section that is mated to the supporting member. You can specify **Top**, **Right**, **Bottom**, or **Left**. Edges of typical section shapes are shown in the figure.

Reflect

Reflects or mirrors the cross-section of the supported member about a plane perpendicular to the supporting member side. For example, when you place a supported member with an angle cross-section using the left edge option and you want the angle facing the other direction.

Surface Frame Connection Properties

X Offset

Specifies an offset to apply in the x-direction.

Y Offset

Specifies an offset to apply in the y-direction.

Z Offset

Specifies an offset to apply in the z-direction.

Coordinate System

Specifies the coordinate system to use for the offset values.

Tangent Frame Connection Properties

Offset Distance

Type the distance (**3**) from the centerline of the supporting member (**2**) to the tangent connection point. This distance is the radius of the circle to which the supported member (**1**) is made tangent.
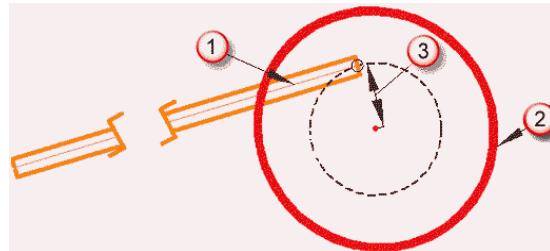
Side

Select **Left** or **Right** to define on which side of the supporting member the supported member is placed. To determine left and right, image you are standing on the start end of the member with your head towards positive global Z looking down the length of the member. Your left and right determine the direction of **Left** and **Right**.

Tangent Plane

Controls the plane on which the tangent circle is drawn. You can select:

- **Horizontal** - The tangent circle is parallel to the X-Y plane (normal to global Z) regardless of the slope of any of the members. For example, you are creating a tangent connection between the sloping green beam and the sloping red column, or between the sloping blue beam and the sloping red column.
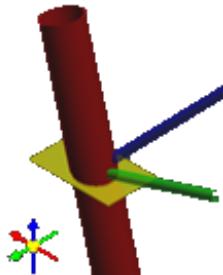
- **Vertical** - The tangent circle is parallel to global Z and is aligned with the centerline of the supported member (green member in the left figure, blue member in the right figure).
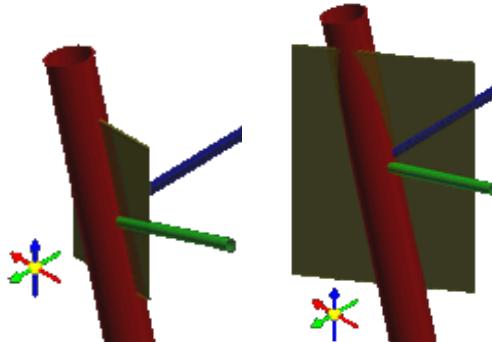
- **Normal to Supporting** - The tangent circle is normal to the supporting member (the sloping red column).
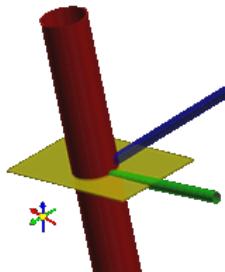
- **In Plane of Supported** - The tangent circle is parallel to the centerline of the supported member and normal to the supporting member centerline (green member in the left figure, blue member in the right figure).

Vertical Corner Brace Frame Connection Properties

X Offset

Specifies the offset to apply in the x-direction.

Y Offset

Specifies the offset to apply in the y-direction.

Z Offset

Specifies the offset to apply in the z-direction.

Coordinate System

Select whether the offset values are defined relative to the global coordinate system or the member's local coordinate system.

Work Point

Specifies the work point location. There are six work point locations that you can select.

1 - Primary Center - Secondary Far Side
2 - Primary Center - Secondary Center
3 - Primary Center - Secondary Near Side
4 - Primary Near Side - Secondary Far Side
5 - Primary Near Side - Secondary Center
6 - Primary Near Side - Secondary Near Side

See Also
Frame Connection Properties Dialog Box (on page 142)

S E C T I O N   1 1

Place Assembly Connection

Places an assembly connection or free end cut at the selected frame connection. Assembly connections define the necessary trimming between member parts and provide for the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections also control cutbacks, copes, notches, bolt holes, weld preparations, and slots.
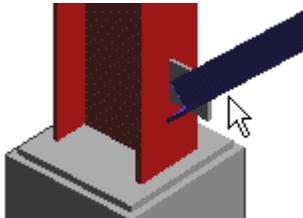
This command places member assembly connections or a free end cut on the member. The software automatically places slab assembly connections when you place the slab.

Member Assembly Connections

Member assembly connections are similar to frame connections, but define the necessary trimming between member parts and the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections control member features including cutbacks, copes, notches, bolt holes, and slots. Whether or not features are placed depends on the member assembly connection type and the geometry of the connection between the members.
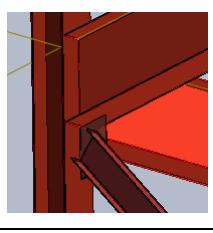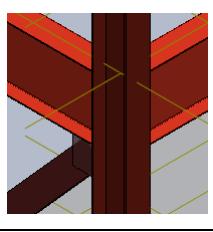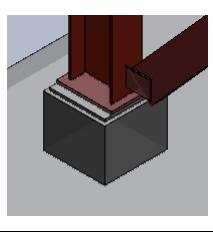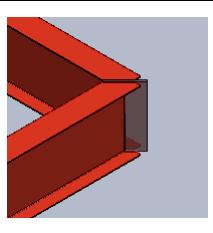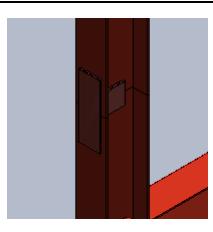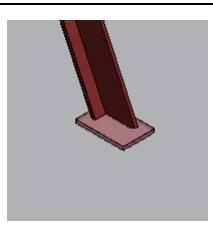
There are several basic assembly connections delivered with the software. You can create your own assembly connections by editing the **StructAssemblyConnections.xls** workbook, and then bulk loading the workbook. For more information on creating your own assembly connections, see the Structure Reference Data Guide.

NOTE   For very complex nodal connections, instead of writing a custom assembly connection, you might want to try *Trim Members* (on page 169) in the Structure task.

The following general assembly connections are delivered with the software:
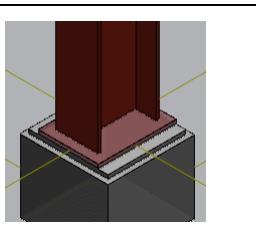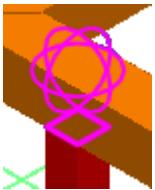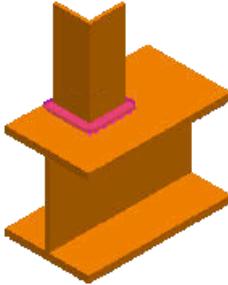
	The base plate assembly connection places a plate at the end of an unsupported member. This assembly connection requires an unsupported frame connection on one member. For example, use this assembly connection to place a base plate at the bottom of a column.

	The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.
	The fitted assembly connection connects two members. This assembly connection requires a frame connection with two members, such as axis, seated, or flush. Examples of this connection include a beam framing into a column or a beam framing into another beam.
	The gusset plate assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. This assembly connection requires a frame connection with two members, such as axis.
	The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection requires a frame connection with two members, such as axis. In addition, the members must be end connected.
	The splice assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The general surface assembly connection connects a member end with a nonmember surface such as a slab, a deck, or a hull plate. The member is cut to surface and a base plate is placed on the member end.

End Cut Assembly Connections

The following end cut assembly connections are delivered with the software:

	The end cut along assembly connection cuts the end of one member to the sides or bottom of another member along the second member's length.
	The end cut bearing plate assembly connection connects the end of a member along the axis of another member. This is typically used to attach the end of a member to the flange of the supporting member. The bearing plate connection works at any angle and provides circular, rectangular, and triangular bearing plates depending on the supported member's shape.
	The end cut miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.
	The end cut split assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The end cut split long box assembly connection is used when the supported and supporting members are perpendicular to each other. The supporting member is cut square to the supported member while an item from the Axis category is placed on the supported member.
	The end cut split seam angle assembly connection is used when the supported and supporting members are: ▪ member axes are collinear to each other ▪ have identical cross-section type and size ▪ have identical cardinal point and rotation values
	The end cut split short box assembly connection is used when the supported and supporting members are perpendicular to each other. The supported member is cut square to the supporting member while an item from the Axis category is placed on the supporting member.

End Cut Assembly Naming Conventions

Each end cut assembly connection name consists of three parts:
[EndCutCase]_[WebCutType]_[FlangeCutType]

where:

- [EndCutCase]

  - **Axis** - Supported member is bounded by the axis curve of the supporting member

  - **LongBox** - End-to-end perpendicular supported member is extended, and an axis end cut is applied to the supporting member

  - **ShortBox** - End-to-end perpendicular supporting member is extended, and an axis end cut is applied to the supported member

  - **Miter** - End-to-end non-collinear

  - **Split** - End-to-end collinear

  - **SeamAngle** - End-to-end collinear where one end is extended

- [WebCutType]

  The WebCutType is used to type of web cut that is to be applied at the top of the web and at the bottom of the web. The following combinations are available (W indicates a straight cut, C indicates a Cope cut, and S indicates a Snipe cut):

  - W#W#

  - W#C#

  - W#S#

  - C#C#

  - C#S#

  - C#W#

  - S#S#

  - S#C#

  - S#W#

- [FlangeCutType]

  The FlangeCutType is used to type of flange cut that is to be applied at the left flange section and at the right flange section. The following combinations are available (W indicates a straight cut, C indicates a cope cut, and S indicates a snipe cut):

  - W#W#

  - W#S#

  - S#S#

  - S#W#

You cannot apply different flange cuts when the supported member has both a top flange and a bottom flange. In these cases, the same flange cut type is applied to both the top and bottom flange.

You cannot apply different web cuts or flange cuts in the end-to-end cases. The end-to-end cases place a web cut and flange cut on both the supported member and the supporting member. For these cases, the same web cut and flange cut are applied to both the supported member and the supporting member.

	1 - Top left flange **2** - Top right flange **3** - Web top **4** - Web bottom **5** - Bottom right flange **6** - Bottom left flange
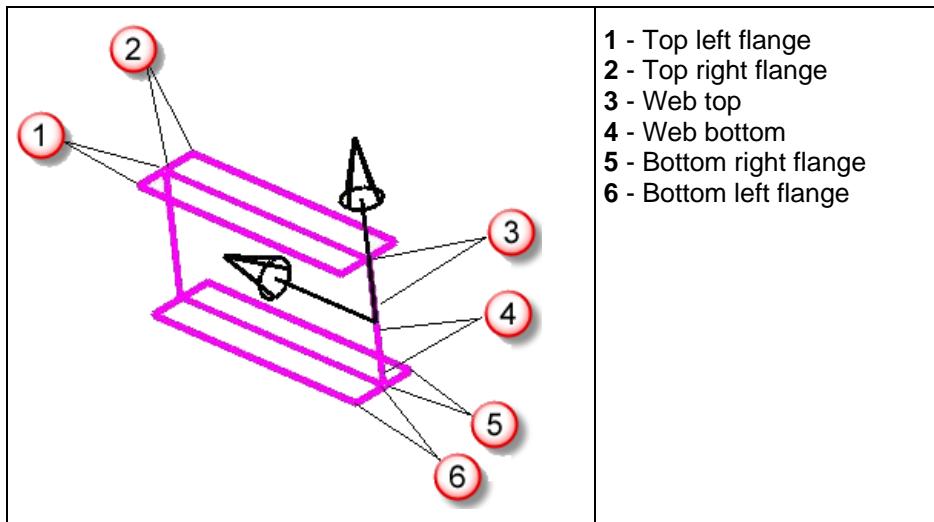
See *Members* in the Structure task for important related information.

Locating Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Member Assembly Connections** in the Structure task, you can locate and select assembly connections for review and editing. Assembly connections are located at the ends of member parts and appear as circles when you move the cursor over them. Any assembly components, such as gusset plates, associated with the assembly connection also highlight.
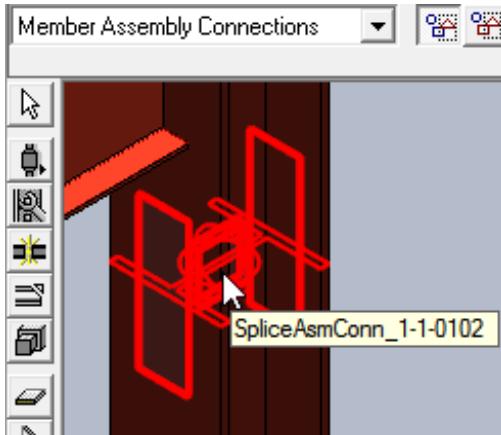
When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties. You cannot measure distances using the assembly connection graphic as an end point when using the **Tools > Measure** command.

Place Assembly Connection Ribbon

Specifies the properties for the assembly connection that you are placing or editing.

Assembly Connection Properties

Activates the **Assembly Connection Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Assembly Connection Properties Dialog Box* (on page 162).

Select Member/Connection

Activated automatically by the software so that you can select the frame connection or the member for which you want to place assembly connections.

Finish

Click to place the assembly connection the model.

Cancel

Rejects the selected object.

Accept

Confirms that the selected members, is the member to place assembly connections for. The software displays in tentative mode the results of the assembly connection.

Condition

Specifies how you want the software to handle existing assembly connections when you try to place a new assembly connection at the same location. Select **Retain existing** to keep the existing assembly connection. Select **Update existing** to replace the existing assembly connection with the new assembly connection.

By Rule

Select to allow the software to select the assembly connection to use based on the selected member parts and their orientation to each other.

Type

Select the assembly connection type to use. If you select **By Rule**, the software determines the correct assembly connection to use based on the geometry between the member parts, and this option is unavailable. If you select **More**, all available assembly connections display from which you can select the assembly connection to use. For more information about assembly connections, see Member Assembly Connections.

Member Part Parent

Select to make the member part the parent object of the assembly connection. Clear this option to specify a generic system, using the **System** box, to be the assembly connection parent. Assembly connections created using member parts as their parents can be edited to have a system parent later if needed.

System

Select the system to which the assembly connection belongs. You can define new systems in the Systems and Specifications task. This option is unavailable if you select **Member Part Parent**. Assembly connections created using a system as their parent can be edited to have a member part as their parent later if needed.

Name

Specify the name of the assembly connection.

Modify Auxiliary Parts Ribbon

Displays the options that you use to add auxiliary parts to the selected assembly connection.

Properties

Activates the **Assembly Connection Properties Dialog Box**, which you use to view and modify the properties of the assembly connection before the assembly connection is committed to the database. For more information, see *Assembly Connection Properties Dialog Box* (on page 162).

Assembly Connection Smart Item

Displays valid smart items for the selected assembly connection.

Add/Delete Auxiliary Parts

Select the parts that can be connected to the selected assembly connection and do not already have an existing assembly connection. Parts that do not have common geometry with the selected assembly connection cannot be selected.

Accept

Adds parts from selected parts to the assembly connection and saves the changes to the database.

Reject

Removes any selected parts from the auxiliary parts list. Only selections made after the last commit are removed.

What do you want to do?

- *Place an assembly connection* (on page 159)
- *Edit assembly connection properties* (on page 160)
- *Delete an assembly connection* (on page 160)
- *Add auxiliary parts to an assembly connection* (on page 160)
- *Edit assembly connection parts* (on page 161)
- *Place free end cuts* (on page 161)
- *Edit free end cut properties* (on page 161)
- *Delete free end cut* (on page 162)

Place an assembly connection

1. Click **Place Assembly Connection** 🖳 on the vertical toolbar.

2. Select the **By Rule** option if you want the software to automatically select the type of assembly connection to place. Clear the **By Rule** option to select the assembly connection type yourself.

3. Select the frame connection nearest the member end to which to apply the assembly connection.
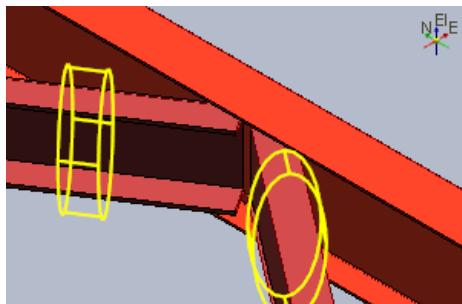
4. If you are selecting the assembly connection type yourself, use the **Type** option.

5. Click **Accept** ✓.

   The software automatically selects the assembly connection and displays the results.
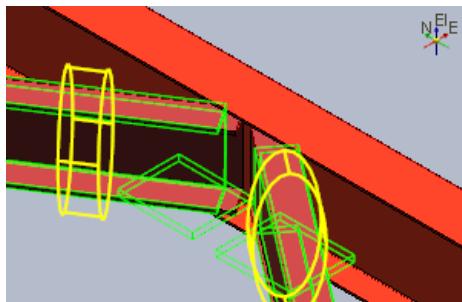
6. Click **Finish**.
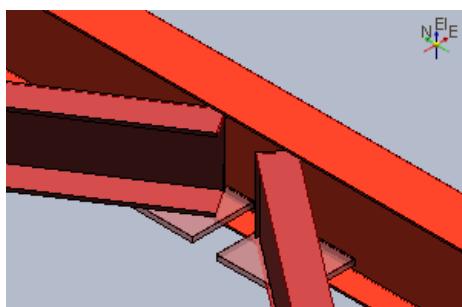
NOTE For more information about the types of assembly connections, see Member Assembly Connections.

Edit assembly connection properties

1. Click **Select** 🔧 on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter**.

3. Select the assembly connection to edit.
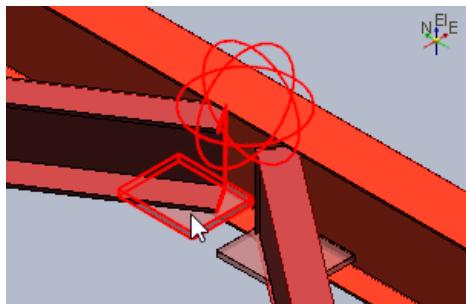
4. Click **Edit > Properties**.

5. Edit the assembly connection properties as needed. For more information, see *Assembly Connection Properties Dialog Box* (on page 162).

Delete an assembly connection

1. Click **Select** 🔧 on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter**.

3. Select the assembly connection to delete.

4. Click **Delete** ✕.

Add auxiliary parts to an assembly connection

This command supports adding parts to an existing assembly connection by selecting an adjacent plate, profile or member parts that have overlapping geometry.

1. Select an existing assembly connection.

2. Click **Select Auxiliary Parts** 🔲 on the ribbon bar.

   NOTE The **Select Auxiliary Parts** 🔲 is enabled for any assembly connection. You can only add valid auxiliary parts to an assembly connection.

3. Click **Accept** ✅ to add all intersecting parts on the selected parts to the assembly connection.

Edit assembly connection parts

1. Select an assembly connection, and click **Select Auxiliary Parts** on the ribbon bar.

2. Select the parts not already connected to the selected assembly connection that have intersecting geometry.

3. To remove a single part, re-select it from the graphic view or **Workspace Explorer**.

4. To remove the auxiliary parts added in the current command execution, click **Reject** on the ribbon bar.

5. Click **Accept** to add all intersecting parts on the selected parts to the assembly connection.

Place free end cuts

1. Click **Place Assembly Connection** on the vertical toolbar.

2. Select the unsupported frame connection nearest the member end to which you want to apply the free end cuts.

   If you select a member part, the software automatically places free end cuts at the unsupported frame connections.

3. Click **Accept.**

   The software automatically selects the free end cuts and displays the results.

4. Click **Finish**.

Edit free end cut properties

1. Click **Select** on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter** box.

3. Select the free end cut to edit.

4. Click **Edit > Properties**.

5. Edit the free end cut properties as needed. For more information, see Free End Cut Properties Dialog Box.

Delete free end cut

1. Click **Select** on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter**.

3. Select the free end cut to delete.

4. Click **Delete** .

Assembly Connection Properties Dialog Box

Specifies the properties for the member assembly connection that you are editing.

Occurrence Tab (Assembly Connection Properties Dialog Box) (on page 162)
Definition Tab (Assembly Connection Properties Dialog Box) (on page 167)
Relationship Tab (on page 106)
Configuration Tab (on page 37)
Notes Tab (on page 39)
Selection Tab (Assembly Connection Properties Dialog Box) (on page 167)

See Also
Edit assembly connection properties (on page 160)

Occurrence Tab (Assembly Connection Properties Dialog Box)

The **Occurrence** tab displays the member assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Name

Displays the name of the assembly connection. The assembly connection name is based on the **Name Rule** selection. To type a new name for the assembly connection, in the **Name Rule** box, select **User Defined**, and then type a name for the assembly connection in the **Name** box.

Name Rule

Specify the naming rule to use to name this assembly connection.

- **Default Name Rule** - Names the assembly connection using this format: <assembly connection>-<location>-<unique index> where <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, GussetPlateAsmConn_1-1-0045.

- **Unique Name Rule** - Names the assembly connection using this format: <parent system name>-<assembly connection>-<location>-<unique index> where <parent system name> is the parent System selected for the assembly connection, <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, Structure System-GussetPlateAsmConn_1-1-0045.

- **User Defined** - Select to specify the assembly connection name yourself in the **Name** box.

System

Select the system to which the assembly connection that you are placing belongs. You can create new systems in the Systems and Specifications task.

Base Plate Assembly Connection Properties

Depth Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension C in the figure.

Width Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension D in the figure.
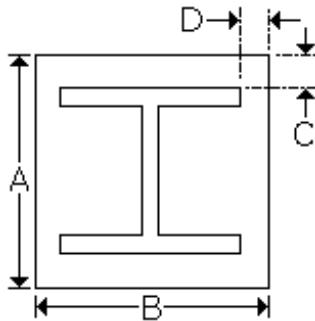
Sizing Rule

Select the sizing rule method for the base plate.

Plate Category

Select the plate category.

Plate Type

Select the plate type.

Miter Assembly Connection Properties

Top Distance

Specifies the distance between the top flange of the member section and the top of the plate.

Bottom Distance

Specifies the distance between the bottom flange of the member section and the bottom of the plate.

Left Distance

Specifies the distance between the left edge of the member section and the left edge of the plate.

Right Distance

Specifies the distance between the right edge of the member section and the right edge of the plate.

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

With Plates

Specifies whether or not a plate is inserted between the member ends. Select **False** to not place the plate. Select **True** to place the plate.

Clearance

Specifies the distance between the member ends.

Sizing Rule

Select the sizing rule method for the base plate.

Slab by Member Boundary

Clearance

Type a clearance distance between the edge of the slab and the member.

Port Face Position

Select the location on the member at which the slab is to stop. You can select the outmost plane, the centerline, or the in-most plane on the member.

Detailed Connection

Select **True** to trim the slab using the boundary member as the cutting edge. Select **False** to not trim the slab.

Offset

Specify the distance between the selected **Port Face Position** and the edge of the slab. A negative value moves the edge into the body of the slab. A positive value moves the edge out from the body of the slab.

Slab by Slab Assembly

Trim Type

Defines the type of trimming.

- **By Whole Slab** - Trim by the slab boundary geometry before trim.

- **By Lateral Face** - Trim by the slab boundary geometry after trim.

- **By Extended Lateral Face** - Trim by the extension of the lateral face of the slab boundary.

Trim Excess Material

Specify whether to trim the exceeding part.

Trim Excess Material Offset

Specify the offset applied on the lateral face used to trim.

Slab Free Edge Assembly

Reference Direction

Select the reference direction for the angle:

Normal

The angle is measured from a vector perpendicular to the slab edge.

- **Horizontal** - The angle is measured from the global XY plane in the model.
- **Vertical** - The angle is measured from the Z-Axis in the model.

Angle

Type a slope for the slab edge represented by the assembly connection. If the slab was place using the **Face Position Top**, the side face rotates about the top slab edge. If the slab was placed using the **Face Position Bottom**, the side face rotates about the bottom slab edge.

Offset

Specify the distance between the selected boundary object and the edge of the slab.

Splice Assembly Connection Properties

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

Clearance

Specifies the distance between the ends of the members.

Splice With

Select the plates that you want to use in the splice.

Web Plate Position

Select a web plate position.

Distance from flange gage line

Specifies the distance from the flange gage line.

Distance from web gage line

Specifies the distance from the web gage line.

Flange Plate Thickness

Specifies the thickness of the flange plates.

Flange Plate Length

Specifies the length of the flange plates.

Flange Plate Width

Specifies the width of the flange plates.

Flange Plate Category

Select the plate category for the flange plates.

Flange Plate Type

Select the plate type for the flange plates.

Web Plate Thickness

Specifies the thickness of the web plates

Web Plate Length

Specifies the length of the web plates.

Web Plate Width

Specifies the width of the web plates.

Web Plate Category

Select the plate category for the web plates.

Web Plate Type

Select the plate type for the web plates.

General Surface Assembly Connection Properties

With Pad

Select **True** to place a steel pad between the member end and the surface. Select **False** to have the member end connect directly to the surface.

Pad Type

Select the shape of the pad.

Sizing Rule

Select whether you want to the software to automatically size the pad or if you want to size the pad.

Offset

Type the distance between the end of the member and the face of the surface.

See Also

Assembly Connection Properties Dialog Box (on page 162)

Definition Tab (Assembly Connection Properties Dialog Box)

The **Definition** tab displays the member assembly connection properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection and then selected the properties command, only the common properties between the selected assembly connections display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also
Assembly Connection Properties Dialog Box (on page 162)

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the member assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.
- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

IMPORTANT

- The available questions, answers, and results vary depending upon the detailed parts associated with the connection.

- The questions in each **Selection** tab represent the default rules delivered with the software. Customized rules may have different questions.

S E C T I O N   1 2

Trim Members

Manually copes and snips member parts based on objects and planes that you specify. Use this command when assembly connections are not sufficient to resolve the interference conflicts due to the position, orientation, and complexity of the intersecting members. You can place a trim and an assembly connection at the same member end. However, the results of the trim might be overwritten by the assembly connection results making the trim redundant. Both the trim and the assembly connection are listed in the **Workspace Explorer** under the member part.

Trim Member Ribbon

Specifies the properties for the trim that you are placing or editing.

Trim Feature Properties

Activates the **Trim Feature Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Trim Feature Properties Dialog Box* (on page 172).

Select Members

Select the members that you want to cut.

Select Cutters

Select the objects to use to cut the members.

Finish

Trims the members using the cutters you have defined.

✕ Cancel

Rejects the selected objects.

✓ Accept

Confirms that the selected objects are the one you want use.

Trim Feature

Select the type of trim you want to place. Click **More** to select from all available trimming types defined in the catalog.

Name

Specify the name of the trim feature.

What do you want to do?

- *Cope a member web* (on page 170)
- *Trim member to surface* (on page 171)
- *Delete a member trim* (on page 171)

Cope a member web

1. Click **Trim Member** ▤ on the vertical toolbar.

2. In the **Trim Feature** box on the ribbon, click **More**.

3. In the tree view, expand **Member End Trim Feature > Web Corner Cope Feature**.

4. Select **Web Cope 1/2in Clearance** in the **Name** column.

5. Click **OK**.

6. Select the member whose web you want to cope.
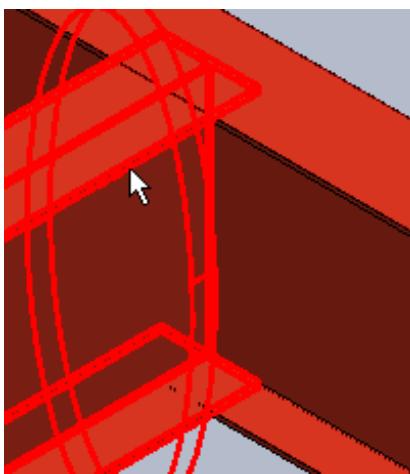
7. Click **Accept** ✓.

8. Select the plane that defines the cut into length of the web.
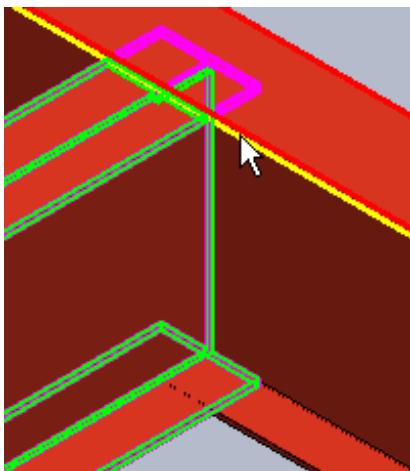
9. Click **Finish**.
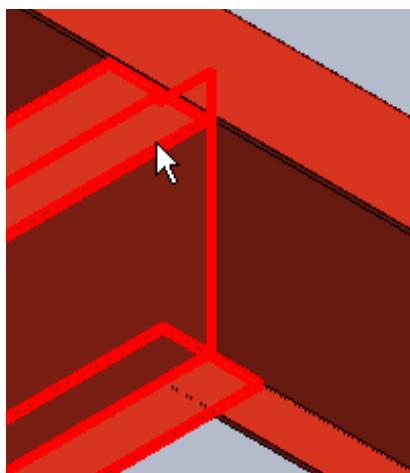
Trim member to surface

1. Click **Trim Member** ▤ on the vertical toolbar.
2. In the **Trim Feature** box on the ribbon, click **More**.
3. In the tree view, expand **Member End Trim Feature > Surface Trim Feature**.
4. Select **SurfaceTrim_1** in the **Name** column.
5. Click **OK**.
6. Select the member to trim.
7. Click **Accept** ✓.
8. Select the surface to trim the member.
9. Click **Finish**.

Delete a member trim

1. Click **Select** ▹ on the vertical toolbar.
2. Select **Structure** in the **Locate Filter**.
3. In the **Workspace Explorer**, expand the member prismatic part branch.
4. Select the trim feature node under the member part branch to delete.
5. Click **Delete** ✕.

NOTE You can also use the **Tools > Select by Filter** command to select the member trims to delete. Define your filter using the **Object Type** tab and the **Structure > Features > Surface Trim Feature** or **Structure > Features > Web Corner Cope Feature** objects.

Trim Feature Properties Dialog Box

Specifies the properties for the trim feature that you are placing.

See Also
Definition Tab (Trim Feature Properties Dialog Box) (on page 172)

Definition Tab (Trim Feature Properties Dialog Box)

The **Definition** tab displays the trim feature properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one trim feature and then selected the properties command, only the common properties between the selected trim features display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also
Trim Feature Properties Dialog Box (on page 172)

Occurrence Tab (Feature Properties Dialog Box)

The **Occurrence** tab displays the feature properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one feature, and then selected the **Properties** command, only the common properties between the selected feature display.

When you view properties for a single feature, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Standard

Name

Displays the name of the feature. The feature name is based on the **Name Rule** selection. If you type a name in this field, the **Name Rule** property updates to **User Defined**.

Name Rule

Specifies the naming rule that you want to use to name this feature. You can select one of the listed rules or select **User Defined** to specify the feature name yourself in the **Name** box.

System

Displays the name of the parent system.

SurfaceTrim_1 Options

Clearance

Displays the clearance value.

Squared End

Specifies whether the end of the member is squared or skewed when the two members are not perpendicular. Set this option to 0 if you want the member end to look like end A in the figure. Set this option to 1 if you want the member end to look like end B in the figure.

Flange Angle

Displays the angle between the normal to the cutting surface and the member axis in the horizontal plane with respect to the member (the flange plane).
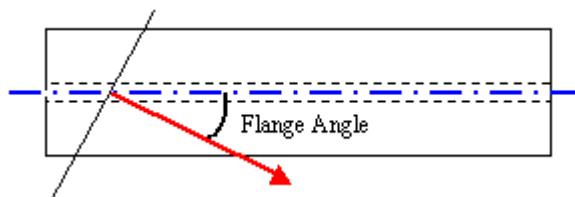
Web Angle

Specifies the angle between the normal to the cutting surface and the member axis in the vertical plane with respect to the member (the web plane).
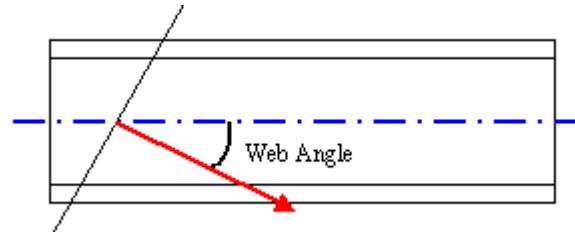
Trim End

Specifies which member end is being trimmed. Type **1** for the start. Type **2** for the end.

Web Cope 1/2 in Clearance Options

Min Cope Length Clearance

Specifies the distance between the flanges. This is dimension B in the figure.

Min Cope Depth Clearance

Specifies the distance between the web and the flange. This is dimension C in the figure.

Rounding Increment

Specifies the value by which to increment the cope dimension. For example, if you set the Increment to 2" (inches), the cope dimension is always divisible by 2 (2", 4", 6", 8", 10", and so on). Similarly, if you set the Increment to 3", the cope dimension is always divisible by 3 (3", 6", 9", 12", and so on). Setting the Increment value to 0 causes the cope to be the exact length needed to clear.

Cope Radius

Specifies the radius of the cope at the intersection of the cope length clearance and cope depth clearance. These are corners A in the figure.
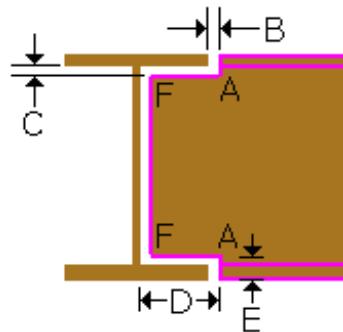
Cope Radius Type

Specifies the type (**Radius** (1), **Chamfer** (2), or **None** (0)) for the cope radius.

Web Radius

Specifies the web radius of the member being coped. These are corners F in the figure.

Web Radius Type

Specifies the type (**Radius** (1), **Chamfer** (2), or **None**) (0) for the web radius.

Cope End

Used for reporting. Leave blank.

Cope Length

Specifies the cope length. This is dimension D in the figure.

Cope Depth

Specifies the cope depth. This is dimension E in the figure.

Cope Edge

Used for reporting. Leave blank.

SECTION 13

Placing Supports from XLS Command

The **Placing Supports from XLS** custom command places supports from an .xls file. The .xls file contains information in the following sheets:

- **Supports** - Includes information about support part numbers, supporting objects, supported objects, and supports configuration.

- **Structure** - Includes supporting object information, such as slabs, members, walls, and structure configuration.

- **Route** - Includes information about pipes, cable trays, HVAC, and supporting object configuration.

The ProgID for **Placing Supports from XLS** is **PlacingSupportsFromXLS,PlacingSupportsFromXLS.PlacingSupportsFromXLS**.

After you add the custom command to the **Add Custom Command** dialog box, the **Place Supports From Excel** dialog box displays:

Select an .xls file, and then click **Process** to read the file for support information and place the supports in Smart 3D. A log file, **PlacingSupportsFromXLSLog_[Date_Time].txt**, is created in your **Temp** folder.

For more information on adding custom commands to Smart 3D, see *Custom Commands* in the *Smart 3D Common User's Guide* and *Smart 3D Project Management User's Guide*.

See also
Place Supports From Excel Dialog Box (on page 176)

Place Supports From Excel Dialog Box

Source Input Spread Sheet

Specifies the .xls file to process for support information. Click **Browse** [...] to open the file directory. By default, the [*Product Folder*]\HangersAndSupports\Testing\Doc folder displays.

Process

Processes the .xls file for supports information.

Cancel

Closes the **Place Supports From Excel** dialog box without processing an .xls file.

Glossary

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

assembly

Unit composed of a collection of parts or other assemblies. Assembly creation consumes the individual part names and provides the unit a unique identification in the fabrication process.

assembly information rule

A program that pieces together individual parts to create a standard support assembly.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

cable hanger

Description of all physical cableway supports.

cableway

Term to describe the volumetric path in a model design through which one or more cables pass from one location in the model to another. Cableway is synonymous with, and is used instead of, raceway or wireway.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

designed support

A customized support that can contain numerous parts and assemblies. You can change and delete the individual parts of a designed support. For example, you can start with a standard support from the catalog and add to it. This type of support is not associative.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

fabricate

To cut, punch, and sub-assemble members in the shop.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

lug (hangers and supports)

A plate with a bolt hole usually welded to the centerline of a pipe. Used to connect the pipe to the other parts of the hanger.

lug and eye piece lifting

Pads used to move structural assemblies.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

part ports

An intelligent connection point on a support part. Ports are used to connect parts together in a physically meaningful manner.

part selection rule

An application that selects a particular part based on the supported and supporting attribute values entered by the user. For example, a part selection rule could select a 6 inch clamp to support a 4 inch pipe.

parts family

Collection of similar parts. Parts families are gathered into a parts library.

parts library

Identified set of data, and possibly programs, that can generate information about a set of parts.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

penetration spool

A spool that can include parts from one or more pipeline systems. The common factor among all the systems is that each pipeline is welded to a common penetration plate.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

reference ports

A location on a beam, pipe, or duct that defines the connection relationship between the parts in the assembly and the beam, pipe, or duct.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specifications

Contracted requirements for the plant.

standard support

A single support object that can be ordered from a manufacturer. The contents of a standard support are an assembly. You cannot change or delete the parts of a standard support. This type of support is associative, meaning that if you change the size of a pipe, for example, the clamp on the pipe changes also.

stud

A bolt, threaded on both ends, used to connect components.

support assembly

A grouping of individual parts, such as beam clamps, rods, and pipe clamps, that comprise a support.

suspended floor

A concrete floor system built above and off the ground.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

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